HyperWorks 12.0 Release Notes

HyperWorks®, The Platform for Innovation™, is built on a foundation of design optimization, performance data management and process automation. HyperWorks is an enterprise simulation solution for rapid design exploration and decision-making. As one of the most comprehensive CAE solutions in the industry, HyperWorks provides a tightly integrated suite of best-in-class tools for modeling, analysis, optimization, visualization, reporting and performance data management. Leveraging a revolutionary "pay-for-use," token-based business model, HyperWorks delivers increased value and flexibility over other software licensing models. Firmly committed to an open-systems philosophy, HyperWorks continues to lead the industry with the broadest interoperability to commercial CAD and CAE solutions.

HyperWorks 12.0 is the new version of Altair's CAE software suite. It includes a large number of new functionalities to support optimization-driven product design and predictive multi-physics analysis, combined with a strong focus on usability and performance.

Modeling and Visualization Capabilities

The modeling and visualization tools in HyperWorks include:

- HyperMesh: A powerful tool for creating and understanding finite-element models
- HyperView: A general-purpose visualization tool for CAE results
- HyperGraph: A user-extensible plotting tool for CAE data
- MotionView: An open tool for understanding and improving multi-body systems
- HyperCrash: Specialized modeling environment for automotive crash and safety
- HyperWorks Collaboration Tools: simulation data organization and management

HyperMesh

HyperMesh 12.0 includes many enhancements to further strengthen core competencies and domain-specific tools and to improve and automate CAE model build and simulation processes. Several new features have been added to enhance HyperMesh's shell and solid meshing capabilities that yield significant reductions in model build times. To further reduce model build time, new connector realizations are available that automate many steps needed to produce a high-quality, fully assembled model. HyperWorks 12.0 sees major gains in workflow efficiency with regard to modeling of composite structures.

Usability

- More information added to the HyperMesh Include View
- Cross-Reference Browser enhancements

Geometry

- Updated interfaces include support for Catia V5-6R2012, UG NX 8.0, ACIS R21 and SOLIDWORKS
- Geometry export now includes STEP and Parasolid
- Solid editing: Cut with plane perpendicular to edge via X-edge selection manipulators

Meshing

- The new MidSurface Mesh tool enables direct extraction of a shell mesh from solid geometry and applies the thicknesses to the corresponding meshed output.
- Thin Solids mesh enables the automatic hexa and/or penta dominant mesh for thin plastic or sheet-metal-type parts

- Mesh extend provides the capability to project an existing mesh to another mesh and will stitch and re-mesh the intersection.
- Mesh imprint provides the ability to take a mesh and transplant the same mesh pattern into a pre-existing mesh.
- Quality Index cleanup tools enable detailed controls on hole and washer manipulations, with the ability to vary diameter, rotate holes interactively (including washers) or set a predetermined dimension.

Composite modeling

- A Fibersim data reader enables users to read results of a draping simulation.
- Catia Composite Design Data (CPD) can be directly imported.
- Fibersim and CPD data can be used in a new process to automatically transfer geometric composites information to finite elements for huge time savings in composite model build time.

HyperBeam

- A shell and solid section sketcher allows users to create, draw and edit shell and solid beam cross sections.
- Interactive section editing enables users to change dimensions of sections and move vertices by "click and drag."

Connectors

- Autopitch: Connector creation on FE at a user-defined pitch distance.
- Complete remeshing for connectors, including spot, seam and bolted realizations.

HyperView and HyperGraph 12.0 bring in new visualization modes, improved performance and a collection of industry tailored solutions, plus many new plotting enhancements.

Industry/Vertical Solutions

- Comprehensive NVH post-processing toolkit
- Crash templates for safety and simplified build plots from the animation client
- Streamline Particle Tracking and Moving mesh support
- Composite ply post-processing with new aggregation modes and filters, shell element forces
- Envelope analysis of linear combinations

HyperView

HyperView is a general-purpose CAE visualization capability in HyperWorks. It can read and display data from a variety of finite-element and multi-body solvers. The highlights of the 12.0 release of HyperView include the following:

- Planar and circular symmetry
- 3D stereoscopic view
- Full tensor visualization

- Network loading of LS-DYNA, and RADIOSS (Bulk Data Format), Nastran results
- Graphics performance improvement in animation and model display manipulation
- Flex body loading, stress contour animation speed-up
- Mesh display modes from the Visualization toolbar on large number of parts

HyperGraph

HyperGraph is a general-purpose CAE plotting capability in HyperWorks. It can read and display data from a variety of sources, including CAE data generated by most finite-element and multi-body solvers as well as data formats used in experimental measurements.

The highlights of the 12.0 release of HyperGraph include the following:

- **Apply Style**: This capability allows you to "apply the style" of the current plot to a selection of plot windows. Any number of plot windows may be selected.
- **Report Layers**: You can control layers created within report templates by renaming or deleting these with a simple right-mouse click. All curves, notes and models in a layer are affected. Layer information is stored in the session file for subsequent reuse.
- **Curve Preview**: You can quickly review curves before adding them to the current session. This is very convenient when large numbers of plots are being manipulated.
- **Metadata Table**: A very simple mechanism for importing metadata from external files into plot notes and headers is now available.
- **Units Support**: All numeric data in HyperGraph can be associated with a set of units and dimensions. Numeric data can be automatically converted from one set of units to another. When plots units are changed, labels are automatically updated.
- **Multiple file selection**: New techniques for simultaneously loading and comparing curves from multiple files or runs are now available. You can use these to quickly understand the meaning of vast quantities of data contained in these files.
- **Customized PowerPoint Publishing**: PowerPoint export has been enhanced to include support for creation of master slides, new tools for image positioning and image resolution independent from the screen resolution.

MotionView

MotionView is a general-purpose multi-body simulation environment in HyperWorks. You can create, analyze, understand and optimize multi-body systems in this environment.

The highlights of the 12.0 release of MotionView include the following:

- **Improved robustness**: This is achieved by providing a new graphical user interface with a more modular architecture, a new assembly container object that allows you to assemble systems easily, more robust entity creation panels, a faster CAD import capability and a major focus on product consistency and bug fixes.
- **Improved ease of use**: Enhancements include a new model browser that has the same look and feel but contains many new usability features, support for new modeling elements, improved messaging, improved solver interface and better tutorials.
- **Model and data management**: This feature, based on HyperWorks Collaboration Tools, allows you to manage and track changes and reuse and share models, subsystems and entity property files.

HyperCrash

HyperCrash is a robust pre-processing environment, specifically designed to automate the creation of high-fidelity models for crash analysis and safety evaluation. It supports both the RADIOSS and LS-DYNA interfaces and their coverage keeps expanding.

HyperCrash 12.0 is an accumulation of 11.0 product updates containing new features, performance improvements, bug fixes and new keywords support for RADIOSS and LS-DYNA.

HyperWorks Collaboration Tools

Simulation data organization and management are available via the HyperWorks Collaboration Tools, which are seamlessly integrated within the HyperWorks Desktop environment and work out-of-the-box with no additional installation or setup required. Collaboration Tools 12.0 includes many new updates over the prior release, including new features and usability enhancements.

- Content labeling to associate groups of contents
- "Manage" files to seamlessly store them in Collaboration Tools during Save As or Export operations
- Browse a PDM system to selectively download and import needed CAD information
- Integration to MotionView client to organize and manage MDL files and other MBD-related contents
- "Recycle Bin" added for full "Windows-like" deletion and restoration or obliteration of contents

Analysis and Optimization

HyperWorks solver technology includes finite-element-based linear and non-linear structural analysis and simulation capabilities (RADIOSS), thermal and fluid analysis (AcuSolve) as well as multi-body simulation (MotionSolve). Combined with design optimization technology and multi-physics capabilities, HyperWorks enables users to drive the product-development process and make reliable decisions based on high-quality results.

With the HyperWorks 12.0 release, Altair continues a successful mix by adding new functionality to accommodate customer needs and latest research results, combined with usability enhancements and performance improvements by leveraging the latest computing technology.

Functionality

- Added functionality to RADIOSS to support automotive powertrain durability analysis and optimization
- New transient heat transfer analysis, including thermal contact in RADIOSS
- Automotive extensions allow co-simulation with third-party components for multi-body analysis (MotionSolve)
- Several new material laws and failure models for explicit and implicit finite-element analysis (RADIOSS)
- Ffowcs-Williams-Hawkings acoustic solver added to the AcuSolve solution suite to predict noise levels at distant microphone locations
- Introduced bidirectional coupling between AcuSolve/AcuTrace for run-time particle tracing
- New manufacturing constraints in OptiStruct for composite materials

Usability

- One-step transfer path analysis (TPA) in RADIOSS for NVH
- Improvements in contact usability; fast and robust solution with less need for tuning (RADIOSS and MotionSolve)
- Tighter integration of AcuSolve within the HyperWorks environment, including incorporation of the documentation into the HyperWorks on-line help system and migration to the InstallAnywere packaging system
- Many ease-of-use improvements, including graphical tools for interpolating results between dissimilar meshes, and simplified utilities for defining turbulent inlet conditions and porosity models
- Significant improvements to AcuConsole database stability and automatic corruption repair

Performance

- Multi-level substructuring solver (AMSES) included in RADIOSS and OptiStruct for large- scale NVH Analysis
- Advanced Mass Scaling (AMS) technology enhancements allows much larger time steps for explicit analysis while keeping the mass constant (RADIOSS)
- GPU support for the implicit solvers in RADIOSS, OptiStruct and AcuSolve
- Performance improvements for rotating bodies in MotionSolve
- Drastic improvements to AcuSolve's scalability for massively parallel applications
- Improved handling of complex CAD models in AcuConsole

Value-Based Business Model

Hosted HyperWorks Units (HHWU)

Altair keeps adding value to customers' HyperWorks licenses through its innovative business model.

- Continuing the legacy, HyperWorks 12.0 introduces Hosted HyperWorks Units (HHWU), the next-generation, cloud-based licensing system. In this new system, HyperWorks can be accessed from anywhere.
- HHWU allows increased portability and scalability and reduced IT burden in maintaining a license server.
- Please contact an Altair representative to learn how to take advantage of the added value and how your current licenses can be moved to the cloud.

HyperWorks On-Demand (HWOD)

- HyperWorks On-Demand is yet another innovative enhancement to the value to HyperWorks.
- Altair now offers access to state of the art hardware on the cloud, utilizing HyperWorks units, through a single web portal for submitting, monitoring and visualizing batch jobs, including remote visualization.
- Contact your Altair representative to learn how to sign up and get activated into the program.

Smart Multiphysics

- Multi-physics and co-simulation licensing has been added, in which the license draw is determined

based just on the number of cores that are used, and effectively lowers costs.

- When supported solvers run on GPUs, each GPU will count for the HWU draw as one CPU-core

Altair Partner Alliance

The Altair Partner Alliance continues to expand its portfolio of third-party technology.

- Applications from more than 30 strategic partners can be directly accessed with HyperWorks Units in the following domains: durability and fatigue, computational fluid dynamics, impact analysis, noise-vibration-harshness, electromagnetic analysis, 1D systems simulation, injection molding simulation, process structure interaction, composite modeling, stress analysis, material libraries, thermal analysis, complexity management, tolerance analysis, casting simulation and multi-body dynamics analysis.

Ideation and Conceptual Design

solidThinking Inspire

solidThinking Inspire allows design engineers, product designers and architects to generate and investigate structurally efficient concepts quickly and easily. Inspire embeds Altair's OptiStruct algorithms, bringing topology optimization to a broader audience. The technology generates the ideal design geometry for a given package space, material properties and loading requirements. Inspire is easy to learn and works with your existing tools to help you design structural parts right the first time, reducing design time, material consumption and product weight.

New, exciting capabilities have been added in this release, including:

- **Sketch Lines, Rectangles, Circles and Arcs**: Build simple design spaces by modifying or creating new solid parts using lines, rectangles, circles and arcs. Sketched geometry can be exported from Inspire in various formats, including Parasolid, STEP, and IGES
- **Boolean Operations**: New solid objects can be created by performing Boolean operations to combine, subtract or intersect sets of objects
- **Frequency Targets:** Design concepts can be optimized to maximize or meet specified frequency targets

solidThinking Evolve

solidThinking Evolve allows industrial designers to develop forms faster than before, using either Windows or Mac, from capturing an initial sketch to exploring styling alternatives and visualizing products with realistic renderings generated in real time. Evolve provides organic surface modeling and parametric control, with NURBS-based surfaces and solids and the unique ConstructionTree history feature. It frees designers from the constraints of engineering-oriented CAD tools, while allowing the export of digital models required by others in the product development process.

Design Exploration

Expand your horizon with HyperWorks design exploration tools, studying the available options in "what-if" scenarios with HyperStudy, understanding system behavior and performing automated complex calculations in HyperMath.

HyperStudy

HyperStudy is a multi-disciplinary design exploration, study and optimization software. Using HyperStudy, users can gain insight into their applications, make better design decisions and optimize the performance, reliability and robustness of their systems.

An entirely re-designed HyperStudy, keeping the overall philosophy of the product the same, incorporates significant UI redesign with communized study approaches, study attributes presented in tables for easy access, new reporting functionality and many more features.

- New features such as study archive, multi-execution and batch mode control
- Extended post-processing that includes multi-plotting options, color maps and reliability plots, among others
- New and improved methods, such as radial basis functions, hybrid method for multi-objective optimization, global response surface method and single loop approach.
- Innovative application of the equivalent static load method for the optimization of geometric and material non-linear problems
- New manufacturing constraints for topology optimization
- A new global search option to avoid being stuck in a local solution

Engineering and Manufacturing Solutions

HyperWorks 12.0 extends the offer for manufacturing and engineering solutions, with new functionalities and an innovative delivery model to better capture knowledge for vertical processes.

HyperWorks Engineering Solutions are now offered as packaged solution offerings (PSO), specialized packages of software and services that support the implementation and utilization of Altair's solutions and technologies, tailored to meet customers' specific requirements. Solutions include:

- Altair PSO for automotive NVH (noise, vibration and harshness) simulation
- Altair PSO for automated reporting

More solutions, such as a PSO for durability simulation, are currently under development. Please contact an Altair representative to learn more about the available Altair PSOs and how to use them with your current HyperWorks licenses.

HyperWorks Manufacturing Solutions, a unified environment for manufacturing process simulation, analysis, and design optimization, include enhanced solutions for sheet-metal forming, metal and polymer extrusion, friction-stir welding, injection molding and forging.

For HyperWorks framework level release notes addressing topics such as hardware support and application licensing, see Software and Hardware Release Notes and Altair Licensing Release Notes.

HyperWorks 12.0 release notes are included for the applications listed below:

HyperWorks Desktop

HyperWorks Desktop	Integrated user environment for modeling and visualization
HyperMesh	Universal finite element pre- and post-processor
MotionView	Multi-body dynamics pre- and post-processor
HyperView	High-performance finite element and mechanical systems post-processor, engineering plotter and data analysis tool
HyperGraph	General-purpose, user-extensible tool for plotting 2D and 3D CAE and experimental data, report generation and publishing

MediaView	Media viewer
TextView	HTML-based text editor
Collaboration Tools	A solution that organizes, manages, and stores CAE and test data throughout the simulation lifecycle
HyperCrash	Specialized modeling environment for automotive crash and safety

HyperWorks Solvers

RADIOSS	A finite element solver for linear and non-linear problems
AcuSolve	A general finite-element computational fluid dynamics (CFD) solver
MotionSolve	A general-purpose, user-extensible, multi-body systems solver
HyperXtrude	A suite of solvers for manufacturing process simulation, applicable to metal extrusion, polymer extrusion, metal rolling, billet forging, friction stir welding, and resin transfer molding.
OptiStruct	Design and optimization software using finite elements and multi-body dynamics

HyperWorks Enterprise

Process Manager	Process automation tool for HyperWorks and third party software; processes can be created with the help of Process Studio.

HyperMath Solutions

HyperMath	Mathematical language for numerical computation
HyperStudy	Integrated optimization, DOE and robust design engine

Manufacturing Solutions

Manufacturing Solutions	A unified environment for manufacturing process simulation, analysis, and design optimization

Engineering Solutions

Engineering Solutions	A unified environment tailored specifically for certain engineering applications

Crash	Specialized modeling environment for automotive crash and safety analysis
AcuConsole	Dedicated pre-processor for CFD analysis with AcuSolve

CAE Results Player

HyperView Player	Plug-in and stand-alone utility to share and visualize 3D CAE models and results

solidThinking

solidThinking Evolve	Comprehensive NURBS-based 3D modeling and rendering environment for industrial design
solidThinking Inspire	Generate and investigate structurally efficient concept designs, bringing topology optimization to a broader audience

CAD and Solver Interfaces

Use the links below to view the HyperWorks 12.0 release notes for CAD and solver interfaces.

RADIOSS (Bulk Data Format) and OptiStruct

RADIOSS (Block Format)

Abaqus

ANSYS

CAD Interfaces

LS-DYNA

Nastran

PAM-CRASH 2G

Samcef

Highlights of the Software and Hardware 12.0 Release

Hardware Platform

- Operating System Additions
- Discontinued Hardware Platform Support

Installation

- New Installation Third Party Software
- Installation and Packaging
- Server/Client Installation
- Installation/Uninstallation
- Invoking

Known Limitations

- Uninstallation
- Installation
- Server/Client Installation
- Help Search
- Resolved Issues

Online Help

- Improved Search Capabilities
- Improved Navigation to Online Help System
- Simplified Help Menu
- HyperStudy Help has been Rewritten
- Updated and Integrated AcuSolve Reference Manuals into the HyperWorks Help System

12.0 Release Notes - Introduction

Various changes have been made to the HyperWorks 12.0 software package including the HyperWorks products, hardware support, installation, invoking, and online help. Changes to the Altair Licensing System are detailed in the Altair Licensing Release Notes.

Back to top

12.0 Release Notes - Hardware Platform

Operating System Additions

Support has been added for:

- Red Hat Enterprise Linux 5.8 and 6.2 (x86_64)
- SUSE Linux Enterprise Desktop 11.0 SP2 (x86-64)
- SUSE Linux Enterprise Server 11.0 SP2 (x86-64)
- Mac OS X 10.7 (x86_64)

Discontinued Hardware Platform Support

Support for the following platforms has been dropped across all HyperWorks applications:

- Linux x86 32-bit (This includes RHEL 5.4, Novell SUSE11.0
- Windows Vista/7 32-bit (x86)
- IBM AIX 5.3 on Power 4/5
- Solaris 10 on 64-bit (x86_64)
- SUSE Linux Enterprise 9 SGI Pro Pack 4 on IA-64 (Itanium)
- Mac OS X 10.6 (64-bit)

Back to top

12.0 Release Notes - Installation

New Installation Third Party Software

HyperWorks installer has been updated to InstallAnywhere 2011SP3 for all supported platforms.

HyperWorks 12.0 installer provides the same installer experience across Linux, Mac OS X, and Windows platforms.

Installation and Packaging

- The HyperWorks software package is split into several independent product packages. It gives user options to download desired packages for installation as needed.
- An SA update can either be installed on an existing 12.0 installation, or can be installed independently without the pre-requisite of a 12.0 base. Multiple product SA updates can now be installed concurrently with or without the 12.0 base.

- AcuSolve is now utilizing the same HyperWorks installation tools and becomes part of the HyperWorks 12.0 package. It is no longer necessary to execute acuSetHW to link the packages after installation.
- The AcuSolve installer now supports installation into directory structures with spaces.
- HyperWorks Help has new book structure and search capability.
- The Demo directory is moved from Help package to the software package.

Server/Client Installation

AcuSolve now supports served application model.

Caching is now available on the client installations that will improve the product invoke time on client machines.

Installation/Uninstallation

- The installation time is greatly reduced.
- The space required for temporary files during installation is decreased.
- The temporary files clean up process is improved.
- The tracking of uninstallation of multiple installations on the same machine is improved.
- Make changes to solver products installation structure.
- Changes were made to the AcuSolve product installation structure to make it more consistent with the rest of the HyperWorks tools. Note that AcuSolve still installs into its own directory within the HyperWorks folder and may be installed with or without a HyperWorks base installation present.
- The WinSxS (Windows side by side) DLL dependency on HyperWorks Desktop applications is no longer needed.
- Focus of desktop windowing during installation and uninstallation on Mac OS X is improved.

Invoking

- Invoking via icon on Mac OS X is now available.
- The icons on Windows and Mac OS X are updated.
- The process for invoking AcuSolve products on Linux has changed. Users may execute the "acusim" script that is located in the altair/scripts directory of the HyperWorks installation to launch a shell with the AcuSolve environment configured. By executing "acusim –h", users can see a list of commonly used AcuSolve invoke commands as well as instructions on how to automatically configure their environment for running AcuSolve at login.

Back to top

12.0 Release Notes - Known Limitations

Uninstallation

When uninstalling a HyperWorks application, all files installed or modified previously will be removed. Backup of any customer files are recommended.

Sometimes, a false warning message at the end of the uninstaller stating that a log file was not able to be removed during the uninstall process may appear. The file/folder will be actually removed a few moments after pressing the "Done" button.

Installation

On Linux, the Help Configuration prompt will add another set of double quotes if the back command is used under console mode. If new input is used, then this issue does not occur.

On Linux, when installing using the console mode, an extra set of quotes will be added to the help browser variable HW_MSG_BROWSER listed in the hwhelp.cfg file (for example : HW_MSG_BROWSER “”/usr/bin/firefox””). Remove the extra set of quotes to make the help work correctly.

Server/Client Installation

For NETSETUP client installs, a UNC path must be used to get to the Server installation path through Windows Explorer or Windows Network.

Help Search

Java plugin is required for help search on Linux.

Resolved Issues

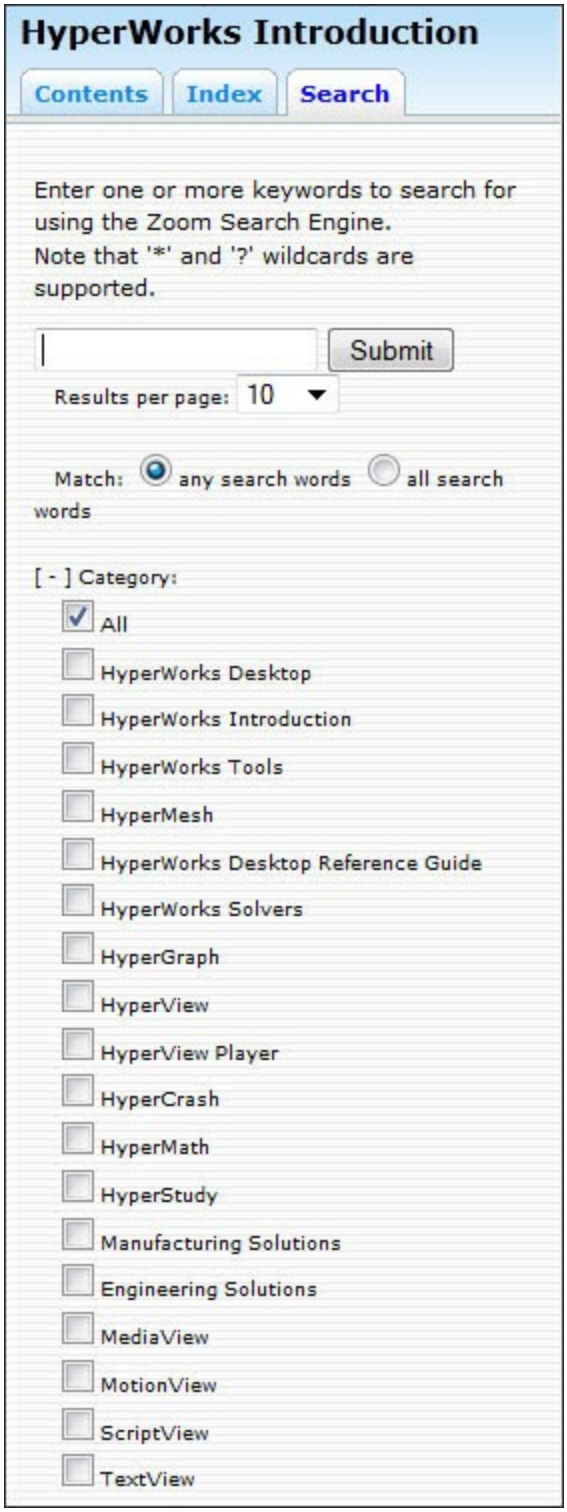
More accurate disk requirement for installation on Linux operating system.

Back to top

12.0 Release Notes - Online Help

Improved Search Capabilities

The Searching capabilities have been greatly improved in 12.0. Instead of having pre-defined groups of user or reference guides to search between, you can select which products and/or guides to search within. There is no limit to how many or how few to include. The results are broken down by product category, so you can see which product has the most number of “hits” on the searched term. In addition, the speed of the search is much faster.



Improved Navigation to Online Help System

HyperWorks 12.0 has eliminated the previous help access points from within the Start menu folder. It has been replaced with a single entry point to the entire help system. This new Welcome Page contains links to all available user guides, reference guides and tutorials. Each user guide and reference guide will have their own link returning you back to the Welcome Page, so that access is always readily available.

Simplified Help Menu

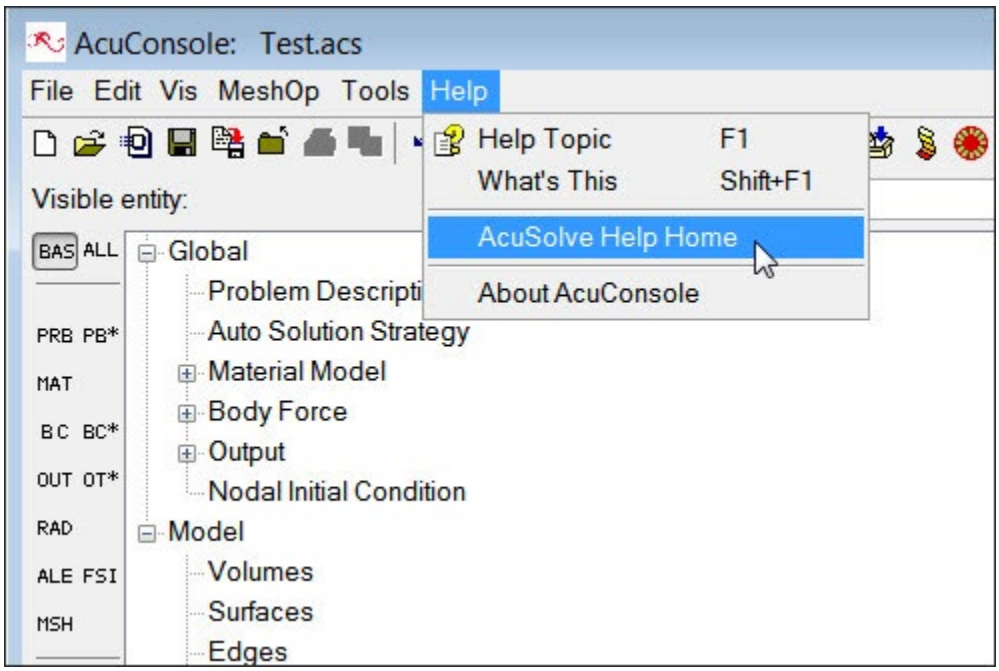
The HyperWorks Desktop applications now contain simplified menu choices from the Help menu. Instead of a long list with preselected options, the menu now contains a link to only HyperWorks Desktop options, and a link back to the Welcome Page.

HyperStudy Help has been Rewritten

The entire HyperStudy online help has been rewritten to reflect the updates to the software. The new online help contains all new content, including updated tutorials and an improved tutorial system.

Updated and Integrated AcuSolve Reference Manuals into the HyperWorks Help System

The full set of AcuSolve reference manuals have been updated and ported to html format and are now integrated with and shipped as part of the AcuSolve installation package. Users have access to the reference manuals through the "Help" drop down menu in AcuConsole, or through the start menu shortcut (Windows only). The pdf reference manuals are also shipped with the installer package and can be found in the /doc directory of the distribution.



AcuConsole Help menu

Back to top

Highlights of the Altair Licensing 12.0 Release

- Inspire and Evolve products included in HyperWorks
- HyperStudy license draw lowered to 25 HWU
- Drop standalone license feature for AcuConsole Parasolid reader
- New multi-physics licensing scheme
- GPU support for HyperWorks Solvers

Contents

HyperWorks Business Model

Application Unit Draw

Solver Unit Draw / Multi-physics and GPU

Special CAD Reader Licenses

Compatibility

Introduction

Same as HyperWorks 11.0, the HyperWorks 12.0 licensing is based on the Altair Licensing System, which provides the unique and flexible Altair HyperWorks business model to our customers. This business model gives full flexibility to access all HyperWorks products, as well as access to partner products through the HyperWorks Partner Alliance (http://www.hyperworksalliance.com). Besides that, the HyperWorks Business model supports large scale computing, design studies, multi-physics and optimization by decaying the license draw with increasing of the usage.

HyperWorks Business Model

- HyperWorks 12.0 licensing is based on HyperWorks Units (HWUs). Each HWU draws 1000 HyperWorks license features.
- The GridWorks Units (GWUs) of HyperWorks 9.0 and 10.0 are replaced by HWUs such that one HWU is equivalent to 100 GWUs.
- The application licensing leveling and stacking behavior remains the same as in the previous release.
- Multi-physics and co-simulation licensing has been added, which determines the license draw just based on the number of cores that are used, and effectively lowers the costs.
- Continuing the legacy, HyperWorks 12.0 introduces Hosted HyperWorks Units (HHWU), the next-generation, cloud-based licensing system. In this new system, HyperWorks can be accessed from anywhere. HHWU allows increased portability and scalability and reduced IT burden in maintaining a license server.

Application Unit Draw

There are a few changes made to the HyperWorks 12.0 product license unit draw:

1. HyperWorks Units required to run HyperStudy are reduced from 38 to 25.
2. solidThinking Inspire and Evolve 9.0 are included in HyperWorks 12.0. It requires 21 HWUs to run Inspire and 10 HWUs to launch Evolve.

Solver Unit Draw

The major license changes for the HyperWorks Solvers (AcuSolve, RADIOSS, MotionSolve, OptiStruct, and HyperXtrude) in V12.0 includes the support of GPU computing and the co-simulation / multi-physics simulation between AcuSolve/RADIOSS and AcuSolve/AcuTrace.

With HyperWorks 12.0 solvers, nVIDIA graphic cards can be used to decrease the runtime. This heterogeneous computing model is particularly suitable for those equation solver dominant types of jobs; for example, nonlinear static analysis on power train structure, or topology optimization on blocky structures, and is available on Linux operating systems.

For the license draw, one GPU is counted as one additional CPU-core. For example, if running a RADIOSS job using 4 cores plus 1 GPU, it will be considered as 5 cores.

When performing a multi-physics or co-simulation run with HyperWorks solvers, the total number of HyperWorks units required is based on the number or CPU-cores plus GPUs that are used in total for this simulation. The HWU draw is determined just like it is done for a single solver run.

For example, AcuSolve on 32 CPU-cores, co-simulating with RADIOSS on 4 cores for a Fluid Structure Interaction (FSI) simulation will be considered in the exact same way as one single job running on 36 CPU-cores.

The base license draw for solver jobs running on 1-4 CPU-cores / GPU is shown in the following table:

HyperWorks Units	Feature Name	Functionality
1	RadiossCheck	RADIOSS (Block Format) Starter RADIOSS (Bulk Data Format) Check-run OptiStruct Check-run
25	RadiossA	RADIOSS (Block Format) Engine
	RadiossB	RADIOSS (Bulk Data Format) OptiStruct (Analysis only)
	AcuSolve	AcuSolve
	AcuView	AcuView
	AcuTrace	AcuTrace
	AcuFwh	AcuFwh
	MotionSolve	MotionSolve
50	OptiStruct	Optimization
	HyperXtrudeFEA	HyperXtrude Solver

For jobs running on more than 4 CPU-cores + GPUs, the following license draw table applies:

Cores + GPU	RadiossA/ RadiossB AcuSolve/ AcuView AcuTrace/ AcuFwh Multi-physics	OptiStruct
1-4	25	50
5-8	30	55
9-16	35	60
17-32	45	70
33-64	55	80
65-128	65	90
129-256	75	100
257-512	85	110
513-1024	95	120
Each Duplication	+10	+10

Note that the Altair Licensing decay function implemented for Radioss, OptiStruct, and AcuSolve products since HyperWorks 11.0 remains the same. The number of the HyperWorks Units draw of a single solver run listed above reduces if more than one HyperWorks solver is running off the same license server.

Special CAD Reader Licenses

In HyperWorks 12.0, the Parasolid reader for AcuConsole no longer requires a separate standalone AcuParaRead license feature.

Compatibility

HyperWorks 12.0 licensing is compatible with HyperWorks 9.1, 10.1, 11.0 and above.

AcuSolve product is fully integrated into HyperWorks 12.0. The AcuSolve 12.0 release can now be launched through the HyperWorks License Manager or via classical ACUSIM licensing via the `-lm_service_type` command line switch.

HyperWorks Desktop 12.0 Release Notes

HyperWorks Desktop is an integrated user environment for modeling and visualization. It can be used to pre-and post-process finite element and multi-body dynamics simulations as well as to manage and visualize test data. The following applications are part of the HyperWorks desktop framework: HyperMesh, MotionView, HyperView, HyperGraph (2D and 3D), MediaView, and TextView. HyperWorks Collaboration tools are fully integrated in this programmable and open environment. ScriptView is fully integrated for TCL scripting and debugging. HyperMesh is available in the Desktop as well as in the traditional standalone mode.

Highlights of the HyperMesh 12.0 Release

HyperMesh API

Part Comparison

*feature_replace

*createintersectionsegments

Connectors

- Add Link Panel
- New Bolt Realization Type
- New Seam and Spot Realization Type
- Nastran Realizations
- Resolved Issues

Geometry

- New Insert Planes Option for Auto Midsurfacing
- Solid Trimming with Plane Perpendicular to an Edge
- Three Node Circle Radius
- New Lines “by attached” Selection Option
- Resolved Issues

Graphics

- Resolved Issues

Meshing

- New Midmeshing Tool
- Mid-thickness Mapping Value Round Off
- Batchmesher
- Thin Solid Meshing
- CFD Tetramesh
- OSSmooth 12.0 Connectivity Detection
- OSSmooth 12.0 Draw Recovery
- OSSmooth 12.0 Ply Shape Interpretation
- OSSmooth 12.0 Geometry Recovery and Remeshing

11.0.130 Release Notes - General

- Browser Include View
- Cross-reference Browser
- Box Trim Utility
- Replace Functionality
- Model Checker
- Assign Dialog
- Check Elements Panel
- FiberSim Reader
- Ply Realization Dialog
- Shortcut Keys
- Deleting Components
- Reordering Components

11.0.130 Release Notes - APIs

- *createlistbypath Panel
- New Commands
- Suite of APIs
- New Interface APIs

11.0.130 Release Notes - Connectors

- Connector Panels
- New Bolt Realization Type for RADIOSS and Nastran
- New Seam Realization Types for RADIOSS
- Autopitch
- Projection Routine
- MAT100 (Hexa) Spot Absorption
- Simple VIP File Import
- Connector Links
- NSM Creation via Connectors
- MCF Import Reader
- Nodes Listed in RBE3 Element Definitions
- Mesh Imprint
- Stiffness Calculation

11.0.130 Release Notes - Geometry

- Geometry Interference Check Tool
- Shortest Distance Measurement Tool
- Resolved Issues

11.0.130 Release Notes - Graphics

- Resolved Issues

11.0.130 Release Notes - Meshing

- Quality Report
- BatchMesher and Autocleanup
- Quality Index Panel
- Mesh Edit Panel
- Isolate Feature
- Tetra Remeshing
- Midmesh Thickness Tool
- Resolved Issues

11.0.130 Release Notes - API

- *createlistbypath panel
- New Commands

11.0.120 Release Notes - General

- Performance Improvements
- Contour Loads
- Visualization Controls Browser
- Catia Reader

11.0.120 Release Notes - APIs

- hm_getsolidmappablestate
- hm_info
- HyperBeam Sections

11.0.120 Release Notes - Connectors

- Non-structural Mass Type Apply Mass Connectors [NSM1/NSML1]
- Links in Connector Browser
- Tolerance used During Connector Creation
- Bolt Realization Types
- hm_ce_checklinentities
- Long Seam Connectors
- Slightly Curved Holes

11.0.120 Release Notes - Geometry

- New Features
- Resolved Issues

11.0.120 Release Notes - HyperBeam

- Add/Delete Vertex for Shell and Solid Sections
- Text Editing on Screen
- Delete Part or Perimeter
- Auto-weld for Shell Sections
- Translation of Shell and Solid Beam Sections
- Export CSV Command Now Available for All Shell Beam Sections
- RADIOSS, Nastran, and OptiStruct Sections Now Display DIM1, DIM2, etc.
- Component Names are Now Placed in Shell Beam Section Part Names
- Command File Support for the HyperBeam GUI
- New Commands to Create Shell and Solid Sections
- Resolved Issues

11.0.120 Release Notes - Meshing

- Map Mid-Mesh Thickness Utility
- Coarsen Mesh Utility

11.0.101 Release Notes - General

- Model Check Tool
- HyperMesh-ScriptView Debugging
- Performance Improvements
- Midsurface Thickness Mapping Utility

11.0.101 Release Notes - API

- New Mass Calculation APIs
- ExtAPI Functions and Structures
- Optimization Entity Support
- *createmark
- *equationhandle and *vectorhandle
- *geommatchtopology
- New Meshing APIs
- *compute_midmesh_thickness
- Resolved Issues

11.0.101 Release Notes - Connectors

- Autopitch
- Feature Angle Recognition for Mesh Transition
- Edge Snapping for Quad Seams
- Seam Direction
- Spot Quad Transition
- Imprint Pattern
- Sliver Elements
- Skip Imprint
- New Seam and Spot Realizations have been Introduced
- Connector Imprint
- Bolt and Stack Panels
- Post Script Handling
- Connector Options Page
- LLINK/PLINK PAM-CRASH Connections
- Seam Connector Panel

11.0.101 Release Notes - Geometry

- Resolved Issues

11.0.101 Release Notes - HyperBeam

- Shell and Solid Section Sketcher
- Interactive Section Editing
- View Controls (Zoom, Fit, Pan)
- Reorient Shell Section
- Preview Unassigned Sections

11.0.101 Release Notes - Meshing

- Map as Circle
- TetraMesh Optimization Utility
- Param File
- Performance Improvements
- OSSmooth Panel

12.0 Release Notes - HyperMesh APIs

Part Comparison

New APIs and functionality (`hm_compare*`) have been added for performing part comparison. Comparisons can be performed between CAD-CAD, CAD-FE and FE-FE. The parts can be located at the same position, translated, rotated, symmetric about a plane, or randomly positioned. Advanced results can be calculated and queried.

---

*feature_replace

`*feature_replace` has been added, to allow for geometrically replacing a selection of surfaces with a second selection of surfaces, reconnecting the new surfaces to the rest of the original surfaces according to the original connections.

---

*createintersectionsegments

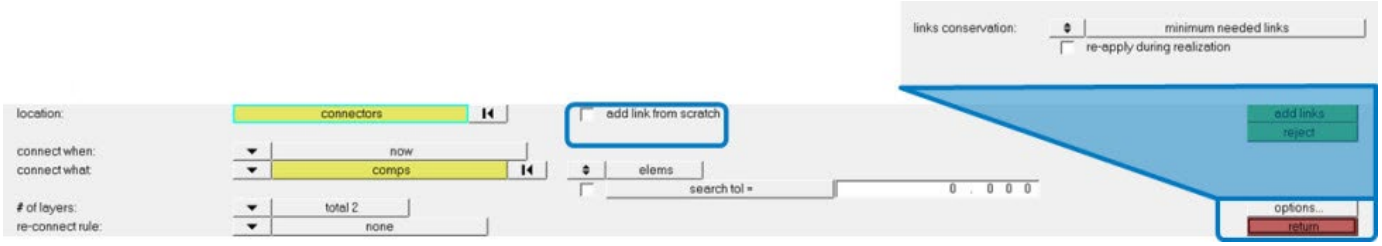
The new command `*createintersectionsegments` can be used to create plot elements showing the intersection segments calculated by `hm_collisioncheck`.

Back to top

12.0 Release Notes - Connectors

Add Link Panel

The Add Link panel has been enhanced. For adding links from this panel the link conservation settings from the connector options are taken into account now. The **Connector Options** button has been added to this panel. Furthermore the option **Add Link from Scratch** has been added. With this option activated, the former existing links are removed before the new links are added.

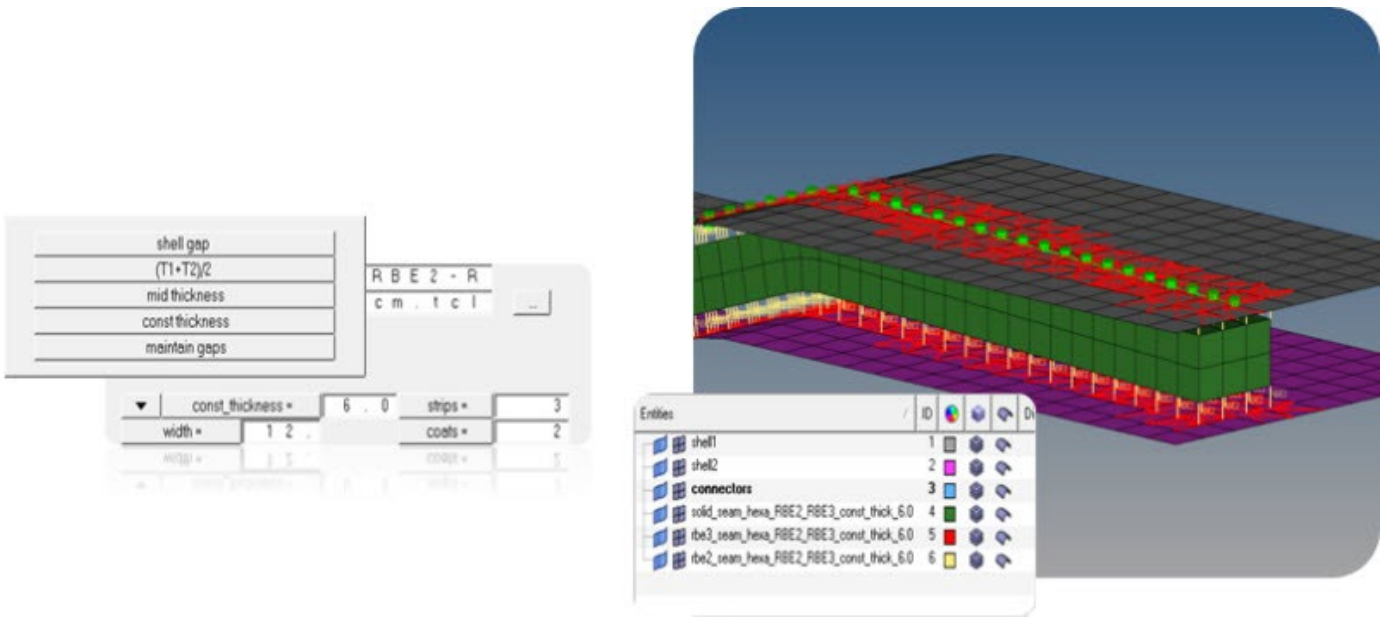


New Bolt Realization Type

For Nastran and OptiStruct a new bolt realization type has been introduced. This bolt is intended to connect one blind and one through hole with each other. The through hole gets an RBE2 spider on both ends, which are connected by a CBAR element. The blind hole gets also two RBE2 spiders, one at the open end and the other underneath the end of the counter bore to represent the thread. Also here both RBE2 elements are connected via a CBAR element. These two constructions are again connected with a CBAR element to each other. The exact geometry depends on the dimensions of the holes.

New Seam and Spot Realization Type

For Nastran and OptiStruct a new realization type for both, spot and seam, has been introduced. The realizations look very similar to the ACMs or hexa adhesive realizations. The only difference is that between the hexa node and the RBE3 node an RBE2 element is introduced. The RBE2 elements bridge in a perpendicular manner the distance between the hexa nodes to the connected shells.



Nastran Realizations

For Nastran realizations using the post script `prop_nastran_acm.tcl` new components, properties and materials have been created per each new realization operation. This has been modified and the creation of those collectors happens based upon the realization type now. For example, if a spot realization of type "ACM (detached-(T1+T2)/2)" is done, the following collectors are created:

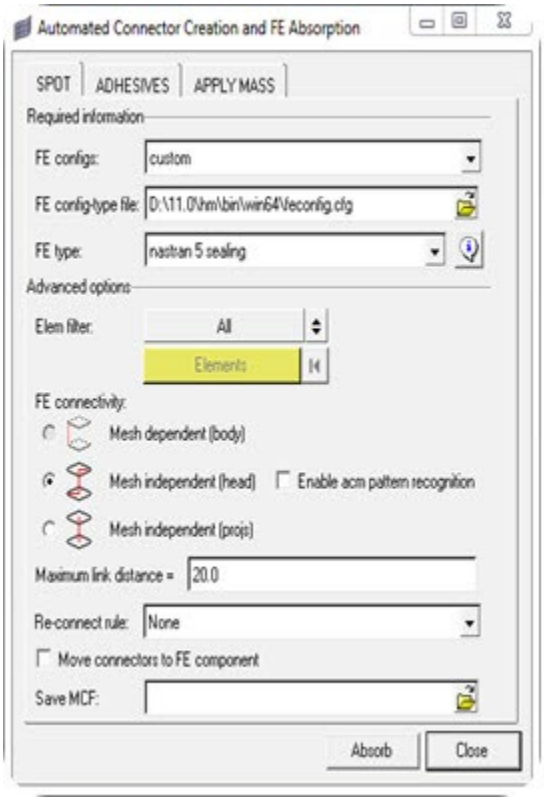
a. Solid elements will be organized into Component by name: **solid_spot_acm_detached**

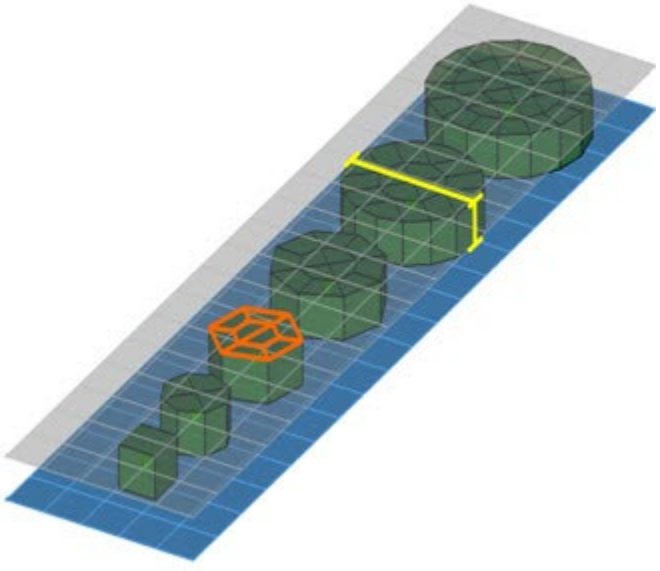
b. RBE3 elements will be organized into Component by name: **rbe3_spot_acm_detached**

c. Property will be created by name: **solid_spot_acm_detached**

d. materials will be created by name: **solid_spot_acm_detached**

New components, properties and materials will only be created when they don't exist. Otherwise the existing collectors are used.





Resolved Issues

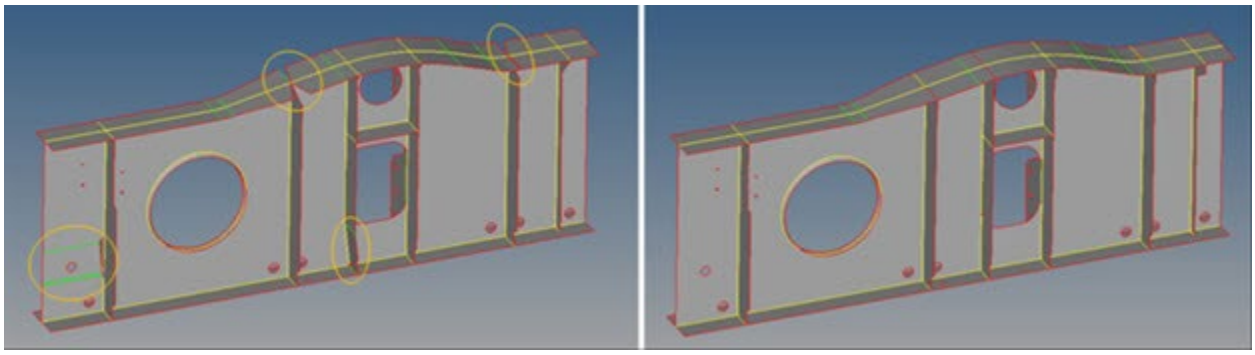
- For various projection methods during link detection and realization only normal projections are allowed. In the past an angle of 5° was considered as normal to cover some of the border cases. This angle has been increased to 10°.
- In certain situations the connector imprint functionality kept the nodes of the former mesh, where the imprint has been applied to. These nodes are invalid and not created anymore.
- Using a seam connector on a surface edge to realize a t-connection between two surfaces led to wrong results, in case of a small gap between the surfaces. This realization case works independent on the surface distance now.
- It is possible now to realize spot connectors in combination with quad transition between two layers of the same component. That led to failed connectors in the past.
- In some cases the realization of area connectors as type hexa adhesive led to flipped hexa elements. This issue was caused by a wrong order of some test points.
- Connector realizations of type CWELD will fail now, if the both parts to be connected have the same property ID.
- In the past during area connector creation per each selected line one area connector has been created even though they formed a continuous line. From such continuous lines it’s possible to create only one connector now. This is optional and can be activated via the checkbox ‘line combine.’

Back to top

12.0 Release Notes - Geometry

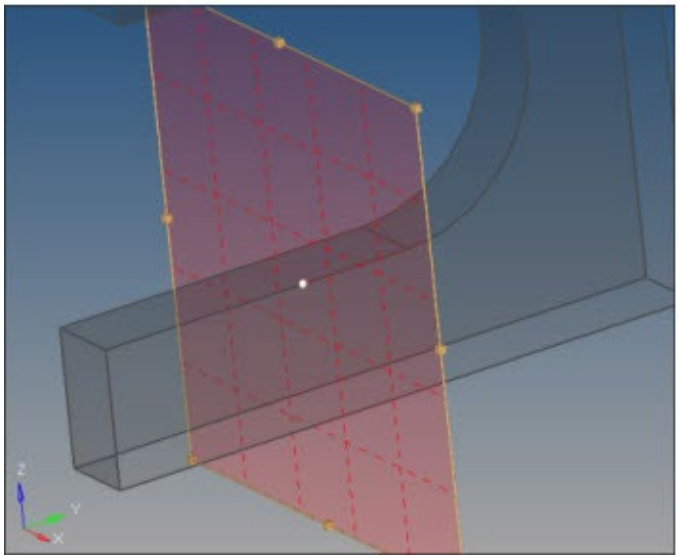
New Insert Planes Option for Auto Midsurfacing

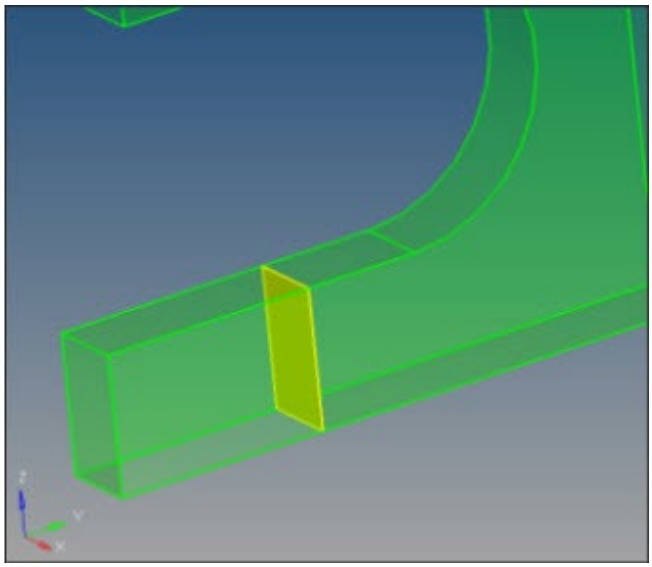
A new “insert plane” option in midsurfacing allows auto recognition of the planar plates and constructing the plate midsurface as a piece of the exact plane. This may significantly improve results for the models with many ribs.



Solid Trimming with Plane Perpendicular to an Edge

This new capability allows you to quickly cut a solid with a plane perpendicular to an edge. The capability utilizes a new graphics manipulator which enables a greater level of control as to where the trim will occur. It enables very localized trim operations and therefore prevents trims occurring on undesired parts of the model.



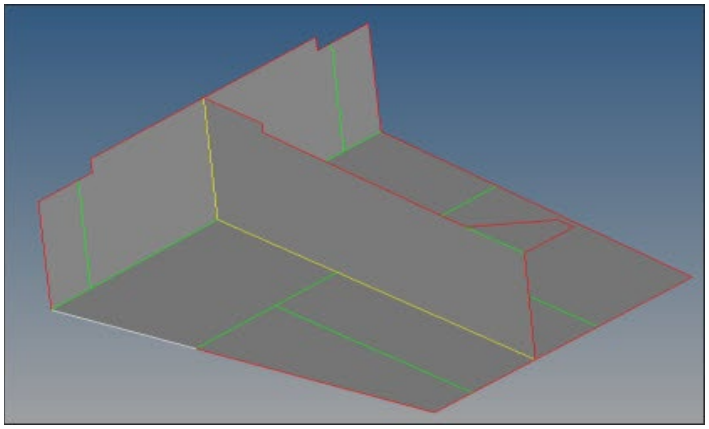


Three Node Circle Radius

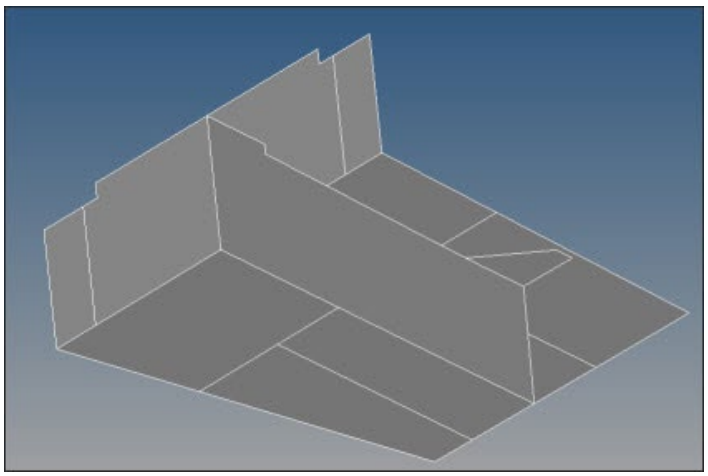
The **Distance** panel now provides an immediate display of three node circle radius without the need to create a center point to know the radius.

New Lines "by attached" Selection Option

Line collectors have now a new **by attached** capability. This applies only to topological lines, not to free lines as they do not have connectivity.



After selecting **by attached**, all lines connected to the first one are selected:



Resolved Issues

- Temporary nodes were disappearing from graphical display when user was using shorter distance tool. This is not the case anymore.
- When using Create in lines component option in surface drag planes, surface was incorrectly created in the current component. Surfaces are now created in the lines component.
- With specific model it could happen that HyperMesh becomes unresponsive during midsurface generation. HyperMesh can now properly generate midsurface from those models without becoming unresponsive.
- When deleting a surface, all masked surface attached to the deleted surfaces would become unmasked. HyperMesh now keeps masked surfaces attached to a deleted surface as masked.
- With specific model it could happen that midsurface was not generate correctly around holes. HyperMesh can now properly generate midsurface around holes from those models.
- Geometry editing panels would not refresh properly when user profile was changed. Geometry editing panels now refresh properly after change of user profile.
- With specific models it could happen that solid editing did not work as expected. Solid editing has been enhanced to be more robust and works in more situations.
- With specific models it could happen that surface editing did not work as expected. Surface editing has been enhanced to be more robust and works in more situations.
- With specific models it could happen that Boolean operations did not work as expected. Boolean operations have been enhanced to be more robust and works in more situations.
- With specific models it could happen that generated midsurface had poor quality. Midsurface have been enhanced to be more robust and generated higher quality surfaces.
- With specific models it could happen that when exporting using Optimize for CAD option the quality of the exported model was poor. Optimize for CAD option has been improved to generate higher quality models.
- Distance panel was reporting distances between nodes incorrectly when using custom axis system. Correct distance is now displayed in the distance panel when using custom axis systems.
- Creating surfaces using Drag along panel would fail on Linux with the following message: “The distance is too small or zero”. Surfaces can now be created using Drag along panel on Linux.
- With specific models it could happen that shortest distance of 0 was reported between solids. Correct distance is now reported as shortest distance.
- With specific models when scaling down then back to the original size some geometry was degenerated. Scaling down then back to the original size do not degenerate geometry anymore.
- When using *reparammark command lines were duplicated but kept the same ID in the HyperMesh database. Each line now has a unique ID in the HyperMesh database.
- With specific models it could happen that points were deleted during surface extension. Points are now correctly kept during surface extension.
- With specific models it could happen that meshing with ExtAPI would return the following message: “API Error: Problems encountered during topology mapping”. Meshing with ExtAPI now works as expected.

Back to top

12.0 Release Notes - Graphics

Resolved Issues

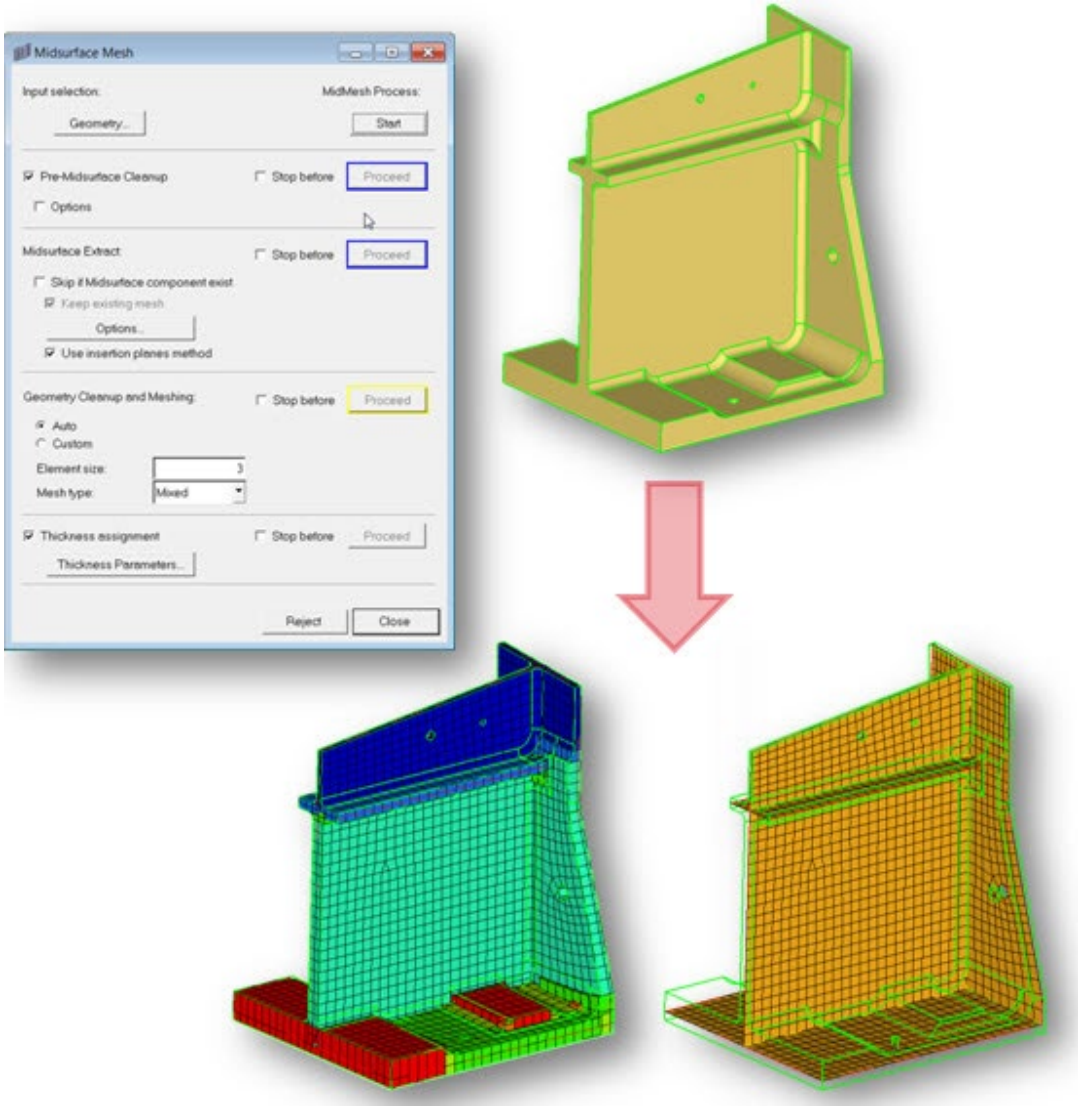
- The rotation center changed for no reason in certain situations on the platform SLED 11.
- There was a slow redraw when rejecting an element deletion.

Back to top

12.0 Release Notes - Meshing

Midmeshing Tool

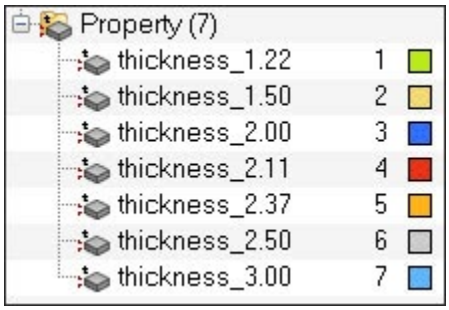
The new midmeshing tool combines three key ingredients, midsurface, QI optimized mesh and midthickness property assignment, into one unified process. The tool is very flexible enabling the user to stop the midmesh process at any stage, make model amendments and then continue the process. There are many detailed operations all of which are found within HyperMesh itself, for example, Pre-Midsurface cleanup houses the options to remove logo's and equivalence the surface edges before midsurface extraction. The tool uses the MidMesh Thickness tool (released in 11.0.120) as the last step in the process. Also, the tool has also integrated the various element visualization options within it enabling for instant review of the midmeshed output.



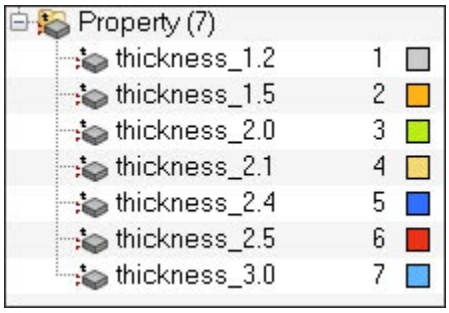
Mid-thickness Mapping Value Round Off

Mid-thickness values can now be rounded off so that assigned thicknesses match ranged interval precision in the **Map Mid-thickness Tool**.

With range interval set to *0.1* and without **Truncate thickness to range interval**:



With range interval set to *0.1* and **Truncate thickness to range interval**:

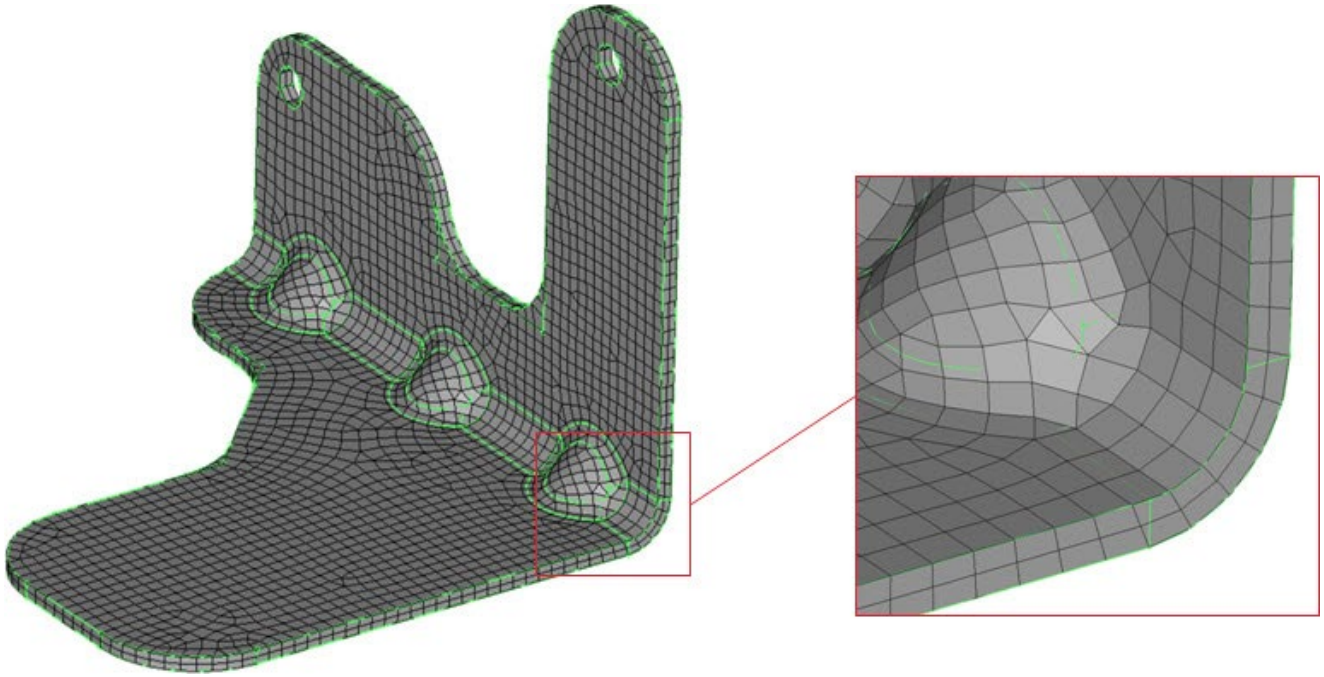


Batchmesher

- Batchmesher has improved recognition, geometry cleanup and meshing for fillets.
- Batchmesher and Autocleanup have new options in the Parameter Editor for handling fillets:
  - Recognize main fillets strips – Recognizes wide/main fillets and tries to preserve them while merging narrow fillets. The side effect is that more narrow fillets close to the minimal element size are not merged.
  - Preserve main edges – Detect and preserve the main edges of wide/main fillets. The side effect is that element quality may worsen due to the addition of more 'hard' features.
- Suppress narrow fillets and surfaces with width: This allows the user to define the maximum width for which narrow fillets and surfaces are suppressed.

Thin Solid Meshing

This new meshing process can create hex/wedge elements for bent sheet metal parts as tetrahedral meshes are not suitable or expensive for analysis. This method automatically detects the top and bottom surface and extrude direction and creates well suited hex/ wedge element combinations.



CFD Tetramesh

- Auto collision detection for boundary layer growth: The boundary layer (BL) reduction step is now included into the BL generation routine. During boundary layer generation a collision check is performed and the boundary layer thickness is adjusted dynamically.
- Advanced parameters for boundary layer generation: Three new options to control the boundary layer growth mechanism have been introduced, including a quality focused and a meshing time focused set of predefined parameters. Selecting the user defined option will give access to several boundary layer parameters which can be adjusted individually for each application.

OSSmooth 12.0 - Connectivity Detection

A new connectivity detection option is available when interpreting the results of a topology optimization. This new option provides a more robust design interpretation process by capturing important material distribution that could otherwise be missed due to the definition of a single density threshold cut-off value.

OSSmooth 12.0 - Draw Recovery

The draw recovery option uses the draw direction input from a topology optimization setup to create straight surface walls in the draw direction, in the isosurface interpretation.

OSSmooth 12.0 - Ply Shape Interpretation

Optimized ply shapes from a composite free sizing optimization can now be interpreted through this option. During the interpretation process, the boundaries of the ply shapes are smoothened, disconnected patches are created as separate ply entities, and options are available to treat very small ply regions. Additionally, after the smoothing and interpretation process is complete, a ready-to-run OptiStruct input deck is generated using the smoothened ply shapes and updated ply definitions, all while preserving the material definition, properties and fiber orientation.

OSSmooth 12.0 - Geometry Recovery and Remeshing

- Improvements to the geometry recovery algorithm result in much cleaner and smoother geometry interpretation of topology results, with fewer number of surfaces.
- Geometry can now be output in PARASOLID and STEP format.
- The remeshing algorithm has been improved to generate a FEA-ready mesh with a mapping of property data, loads and boundary conditions, setting up the optimized model for a quick reanalysis.

Back to top

11.0.130 Release Notes - General

Browser Include View

Added three new columns within the browser include view to set the export state, include path, and include type of one or more includes. This is available for all applicable user profiles.

Cross-reference Browser

Display control within the cross-reference (XRef) browser is now available via the Show/Hide/Isolate entries in the context menu. Review is also added to the context menu to set the review state of the selected entity.

Box Trim Utility

The Box Trim utility in the LS-DYNA user profile has undergone significant updates to enable the box trim to remove unused entities and automatically update entities which pass through the box trim – the objective is to ensure a run ready deck post box trim. The same utility in the Nastran user profile has had some additional updates also.

Replace Functionality

Added "no to all" option in the Replace functionality.

Model Checker

Model Checker has been updated to detect and remove unused beamsections for OptiStruct, RADIOSS (Bulk Data), Nastran and LS-DYNA user profiles.

Assign Dialog

Added Name, Name(ID), ID list display options to the Assign dialog for all user profiles.

Check Elements Panel

Added the ability to find free nodes via the Check Elements:1D panel, free 1-d nodes.

FiberSim Reader

New FiberSim reader has been added to read Composites design data from FiberSim HDF5 files. The FiberSim reader selection is available in ***Import*** > ***Geometry*** > ***File Type***. The **file type** drop down menu has a new option called **FiberSim** to read FiberSim HDF5 file formats.

Currently the FiberSim interface is supported only for RADIOSS (Bulk) and Nastran interfaces, but will be supported in other interfaces in future releases.

Ply Realization Dialog

A new option called **FiberSim Drape Map by Proximity Method** is added to **Projection Options** in the **Ply Realization** dialog to extract elements for ply boundaries, and to map drape data to the target mesh.

Using this method, if the element centroid projected along every element normal lies within the FiberSim ply definition (in terms of triangulation on a ply table), it is associated with this ply upon realization. After extracting the elements for ply, the FiberSim drape data is mapped to new ply elements within HyperMesh. A drape card image is automatically assigned to an associated table of each ply in the RADIOSS (Bulk Data) interface in HyperMesh during the ply realization process.

Shortcut Keys

Shortcut keys Selector (S), Show/Hide (U) and Isolate (I) are re-enabled in 11.0.130.

Deleting Components

In previous versions, users were unable to delete components in the browser with large ID’s. This has been resolved.

Reordering Components

In previous versions, if the user reordered components then the entity name in the browser was incorrect.

Back to top

11.0.130 Release Notes - APIs

*createlistbypath Panel

`*createlistbypath` panel added, which puts the panel list collector in the “by path” mode automatically.

New Commands

`hm_getincrementalname` for generating a unique entity name across multiple entity types.

`hm_measureshortestdistance` and `hm_measureshortestdistance2` for finding and measuring the shortest distance and location between nodes, lines, points, surfaces and solids.

Suite of APIs

Suite of APIs for geom-geom, geom-FE and FE-FE comparison generation and query.

New Interface APIs

New interference APIs `hm_collisioncheck`, `hm_collisionend`, `hm_collisionentitycreate` and `hm_collisioninit` APIs for performing geometry interference checks. These will be updated in the future to also support FE and to replace the existing FE-penetration APIs.

Back to top

11.0.130 Release Notes - Connectors

Connector Panels

The main connector panels **Spot**, **Seam**, **Bolt** and **Area** have been completely reworked to make them as consistent as possible. All of those panels are organized into three clearly and topically subdivided columns. The left panel column contains everything related to the connector creation and link detection, the middle panel column contains everything related to the connector realization type and the post script or property assignment and the right panel column contains everything related to the final connection to the link entities.

The newly introduced **Connector Options** page has been further enhanced. It contains general options, but also specific options per connector type now.

A new option **consider shell thickness and offset for hexa positioning** opton has been introduced on the general Connector Options page. With this option activated (default) the hexas are not just positioned in the exact middle between the two shells, they are positioned in the middle of the theoretical air gap now. Shell gap realization types work exactly the same as before.

New Bolt Realization Type for RADIOSS and Nastran

A new bolt realization type for RADIOSS has been introduced - bolt (2 cylinder rigid). In contrast to other cylinder bolt types, where just the nodes inside the cylinder are considered for the connection, this type is combined with the hole detection mechanism. If a hole can be detected inside the cylinder just the hole is connected, else all nodes inside the defined cylinder. For this bolt type two different diameters for L1 and L2 can be defined.

A new bolt realization type for Nastran has been introduced - clip (washer nodes). In difference to the original clip realization this type connects exclusively the washer nodes.

New Seam Realization Types for RADIOSS

Two new seam realization types for RADIOSS have been introduced - type2 (spring multiple row) and type2 (spring single row). Both types create the exact same connection as done by the spot realization type2 (spring). The only difference is that with these seam realization types, many of those single spots can be performed at once distributed in a very regular pattern. So you can define along one seam multiple parallel lines along those single connections are distributed with a certain distance to each other.

Autopitch

The newly implemented Autopitch functionality has further been enhanced and flanges can be detected now, which weren’t detected before.

Projection Routine

The projection routine used for hexa creation during spot or seam realization has been greatly

improved. In particular, the projection onto solids works much better now.

A new option for seam hexa (adhesive) ‘maintain gap’ has been introduced. This option manages the hexas to be created in between two shells like with the normal shell gap option, but keeps a certain gap distance between shell and hexa.

MAT100 (Hexa) Spot Absorption

The MAT100 (hexa) spot absorption has been enhanced. In the past the absorption was only possible, if the hexas were attached to the shells by plot elements. This is not necessary anymore. Furthermore the hexa pattern as well as the diameter and the tolerance are determined now. In principle this capability is available for all hexa spots when selection ‘projs’ instead of ‘head’ in the absorption window. For those cases it is recommended to select only the real spot hexas to be absorbed because not all situations can be absorbed reliably.

Simple VIP File Import

The simple VIP file import has been enhanced, and the export is also available now.

Connector Links

The strategy to conserve more links on the connector than necessary has been changed. For spots and seams we can define a number for extra links on the appropriate connector options page. For a spot the defined extra links plus the number of layers is the maximum number of links, which can be stored on a connector. For seams this number has to be multiplied by the number of testpoints. The default setting is ‘minimum needed links‘, which reduces the links to those, which are necessary for a successful realization. Especially the PamCrash PLINK and LLINK use the extra links.

NSM Creation via Connectors

The newly implemented NSM creation via connectors has been enhanced. In the former version you could only create NSM of type ELEMENTS. Unless the circumstances are clear, you can create NSM of type PSHELL now too. The absorption and the apply mass function allow further settings as well.

MCF Import Reader

Under certain circumstances the MCF import reader didn’t recognize differences between the different connector types and all connectors were imported as spots.

Nodes Listed in RBE3 Element Definitions

In seldom cases nodes were double listed in RBE3 element definitions, which were created during the penta (mig) realization.

Mesh Imprint

The mesh imprint functionality has been improved. In the past 1D connections in the area of mesh imprint were not considered and disconnected.

Stiffness Calculation

There was a small mistake in the stiffness calculation of for the HiLock realization.

Back to top

11.0.130 Release Notes - Geometry

Geometry Interference Check Tool

A new Geometry Interference Check tool is available which will find penetrations and intersections between geometric surfaces and solids. The tool is located in the ***Geometry*** > ***Check*** > ***Surfaces/ Solids*** > ***Interference*** menu. The tool enables you to perform geometric checking of penetration, intersection or both penetration and intersection. The tool also has some display control to isolate failed components or surfaces, and the tool allows you to save surfaces or components that failed.

Shortest Distance Measurement Tool

A new Shortest Distance measurement tool is available for calculating the shortest distance between selected nodes, points, lines, surfaces and solids. The tool is located in the ***Geometry*** > ***Check*** > ***nodes/points/lines/surfaces/solids*** > ***Shortest Distance*** menu.

Resolved Issues

- Fixed incorrect stitching when trimming during surface extension.
- Improved accuracy of ruled surfaces.
- Correctly consider bias intensity when interpolating nodes.
- Fixed crash when creating surfaces from multiple crossing lines.
- Improved autocleanup results.
- Prevent PLOTEL elements to be removed during autocleanup.
- Removal of surface and solids holes in one pass during autocleanup.
- Fixed crashes with some models during autocleanup.
- Prevent surfaces deletion during flange recognition in autocleanup.
- Fixed crashes with some models during defeaturing.
- Prevent solid deletion during solid edition with some models.
- Improved surface trimming result for some models.
- Correctly consider z-offset data in midsurface thickness utility when assigning thickness to properties on components.
- Correctly assign thickness information when creating midsurface using surface pair option.
- Fixed inconsistencies in results when splitting lines.
- Fixed crash during point replacement with some models.
- Improved stitching during extension of adjacent surfaces.
- Improved trimming of surfaces with planes.
- Fixed crash during remeshing after surface extension with some models.

Back to top

11.0.130 Release Notes - Graphics

Resolved Issues

- Group elements were not highlighted correctly during direction changing.
- Some background color problems have been corrected.
- When using the geometry color mode mappable the surface editing caused a reproducible crash.
- In certain constellations the graphics turned into a solid blue and covered the model.
- After specific actions certain elements could not be selected anymore.
- The hidden line mode caused a crash when pyramids where present.
- The fillet center crosses were disappeared during model rotation, which made it impossible to select them during fillet creation.
- For certain models the highlighting of selected surfaces was pushed completely into the foreground, even if other surfaces were on top of them.

Back to top

11.0.130 Release Notes - Meshing

Quality Report

A new **Quality Report** is available within HyperMesh and Batchmesher. The report is an HTML based report that enables a detailed breakdown of the component(s) quality based on a predefined criteria file. Within HyperMesh, the report can be found at ***Mesh*** > ***Check*** > ***Elements*** > ***Quality Report*** and within the Batchmesher the report is available via the ***Generate Quality Report*** checkbox in the **Run Setup** tab.

BatchMesher and Autocleanup

Batchmesher has improved feature recognition and capturing of fillets, beads, complex beads, bolt holes and plateaus.

Apply optimized smoothing: Apply failed features cleanup is a new option to ignore geometry features when fixing elements which failed the quality threshold. The option is on by default.

Batchmesher and Autocleanup have new options in the Parameter Editor:

- Target radius - holes in the range are adjusted to have the specified target radius. The radius can be specified as an exact value (e.g. 5.0), or as an expression based on the original radius (e.g. radius*1.1, radius-0.5, radius+0.5).
- Within other options, there is an option to delete duplicated surfaces within a given tolerance.

Quality Index Panel

Within the **Quality Index** panel, new cleanup tools have been added.

- Place node – along surface, allow movement out of boundary
- Place node – normal to surface
- Modify hole & washer – radial, angular, radial & angular, circumferential – allows complete interactive control of hole treatments.

- Node optimize – normal to surface
- Node optimize – along & normal to surface

Mesh Edit Panel

A new **Mesh Edit** panel enables the automation of imprinting one mesh into another and also provides the ability to extend one mesh to another.

- **Imprint** allows you to select a source/destination mesh and define a direction, the result is one mesh imprinted into the other. This is powerful functionality which satisfies many use cases, especially for node matching welded model setup or node matching contact setup, or, just a quick efficient way of ensuring equivalenced meshes after imprint.
- **Extend** allows a node selection to be extended to selected elements/components and automatically remesh around the intersecting junction. There is additional control to remesh the extended elements, or, retain the elements as a simple extension with no remesh. Advanced controls to remesh surrounding layers are also available.

Isolate Feature

In the **Tetra Optimization** panel, a **Show Failed** button as been included to show only the failed elements.

Tetra Remeshing

New options to set outer boundary shells as fix, swap or remesh have been added.

Midmesh Thickness Tool

Two additional controls have been added to the Midmesh thickness tool:

- Max Midmesh/Solid angle - The maximum angle that the mid-mesh makes with the solid, beyond which the estimated thickness will be ignored and corrected.
- Max thickness gradient - The maximum change in thickness allowed across two adjacent measurement locations, as a factor of the distance between the locations.

Resolved Issues

- Resolved a segmentation error if there are no solids selected in the Midmesh Thickness tool.
- Updated the Autopitch functionality to support additional use cases; there were cases where connectors were not created on some portion of flanges in some specific models.
- For some specific cases, automesh remesh elements with feature edges was failing. This has been fixed.

Back to top

11.0.130 Release Notes - API

***createlistbypath panel**

`*createlistbypath` panel added, which puts the panel list collector in the “by path” mode automatically.

New Commands

New command `hm_getincrementalname` for generating a unique entity name across multiple entity types.

`hm_measureshortestdistance` and `hm_measureshortestdistance2` for finding and measuring the shortest distance and location between nodes, lines, points, surfaces and solids.

Suite of APIs for geom-geom, geom-FE and FE-FE comparison generation and query.

New interference APIs `hm_collisioncheck`, `hm_collisionend`, `hm_collisionentitycreate` and `hm_collisioninit` APIs for performing geometry interference checks. These will be updated in the future to also support FE and to replace the existing FE-penetration APIs.

Back to top

11.0.120 Release Notes - General

Performance Improvements

Major performance improvements when importing a solver deck over a network for HyperMesh and HyperView OptiStruct, Abaqus, Nastran, and LS-DYNA readers. Performance gain is subject to network conditions.

- Importing a single solver deck over the network expect up to 40% improvement.
- Importing a solver deck with includes over the network expect up to 73% improvement

Contour Loads

Renamed the BCs Contour utility to Contour Loads.

Added a new data type **Pressure - Signed MAG** to the Contour Loads utility that considers if the resultant is in the same direction as the element normal (positive sign) or in the opposite direction (negative sign).

Visualization Controls Browser

Tags visualization support has been added to the Visualization Controls browser, and visualization options have been added to the Tags panel.

Catia Reader

Catia reader has been enhanced to read Composites Parts Design data (CPD) from Catia files. Related to this, the context sensitive menu of the Model Browser holds new functionality:

- **Realize ply**: Geometric ply shape information can be transformed into its FE representation. Elements belonging to a ply shape can be identified based on three projection rules described next in these release notes.
- **Realize laminate**: Laminates can be realized into a FE model. Properties are created and assigned to elements to translate the generic model definition into a solver specific one.
- **Review `by thickness difference**: For continuum shells the difference between the thickness defined on the property and the true element thickness is shown as contour plot. The elemental thickness is defined as average of the distance between the nodes on the bottom and top face in the stacking direction.

The new functionality has been added for RADIOSS (Bulk), Abaqus and Nastran interfaces, but can be added to other interfaces in further releases. Functionality related to continuum shells is supported for Abaqus interface only.

Back to top

11.0.120 Release Notes - APIs

hm_getsolidmappablestate

`hm_getsolidmappablestate` added for querying the mappable state of solid geometry.

---

hm_info

Added new options **elementcolormode**, **elementinternaledgesmode**, **picktolerance**, **rotationangle**, **surfaceuvlines** and **topologydisplaymode**.

HyperBeam Sections

Added new APIs for creating/editing HyperBeam sections

Back to top

11.0.120 Release Notes - Connectors

Non-structural Mass Type Apply Mass Connectors [NSM1/NSML1]

- NSM type connectors control HyperMesh groups with NSM1 and NSML1 card images.
- FE absorb is implemented for NSM type apply mass connectors.

Links in Connector Browser

Links can be updated to NA in the **Connector Browser** now. This became necessary due to the capability to create bolt connectors with unlimited layers.

Tolerance used During Connector Creation

The tolerance used during connector creation is stored on the connector now. In the past only the tolerance used during connector realization has been stored. Since mostly the same tolerances are used for link detection and realization, this has been updated.

Bolt Realization Types

The projection logic for bolt realization types defined by cylinder definition (L1/L2, diameter) has been updated. There is no longer a need to place the connector somewhere in the middle between the links.

For bolt realization types defined by cylinder definition (L1/L2, diameter) a further attribute has been added, cylinder diameter factor. This factor multiplies the given diameter and is used to guarantee that the nodes of hole wall fall into the defined cylinder. Cylinder diameter factor as well as diameter are optionally displayed in the Connector Browser and can be updated from there.

hm_ce_checklinentities

The command `hm_ce_checklinentities` works also for assembly names now.

Long Seam Connectors

For some very long seam connectors the link detection worked wrong and some obvious links weren't detected. This issue has been resolved and also the limitation for the maximum number of links, which can be stored on a seam connector has been removed.

Slightly Curved Holes

In a few cases slightly curved holes weren't accepted as true holes, which caused failed connectors. This issue has been resolved.

Back to top

11.0.120 Release Notes - Geometry

New Features

- New option for choosing stitching mode when creating surfaces.
- Added thickness based stitch tol option for midsurfacing.

Resolved Issues

- Continued robustness and accuracy improvements for surface and solid editing.
- Continued robustness and accuracy improvements for midsurface generation.
- Continued robustness and accuracy improvements for defeaturing operations.

Back to top

11.0.120 Release Notes - HyperBeam

Add/Delete Vertex for Shell and Solid Sections

This mode is accessed by pressing the second to last button in the toolbar. This button is only active for shell and solid beam sections.

Clicking this button launches into add and delete vertex mode.

In this mode, left clicking on or near any part (for a shell) or perimeter (for a solid) will create a new vertex on the nearest part or perimeter. The part or perimeter will be diverted to now run through the new vertex. If a clicked point is nearer to the end of a part than a segment of it, the part will be extended.

In this mode, right clicking any vertex will delete the vertex.

Text Editing on Screen

This mode is accessed by clicking the third to last button in the toolbar and is the same as interactive dimension changing, vertex moving, and thickness altering.

In this mode, double clicking on a dimension or a thickness, which are represented on screen as text, will allow create a box around the text and allow you to edit it. The only available options to edit the text are backspace to delete characters, the ten number keys, and the period. Exponential notation is not allowed.

Delete Part or Perimeter

This mode is accessed by pressing the last button in the toolbar. This button is only active for shell and solid beam sections.

Clicking this button launches into delete part or perimeter mode.

In this mode, left clicking on or near any part (for a shell) or perimeter (for a solid) will delete the nearest part or perimeter.

Auto-weld for Shell Sections

This functionality can be accessed in the HyperBeam GUI by right clicking a shell beam section in the browser and selecting "Auto-weld." It can also be accessed in the HyperBeam panel using the "section utils" radio button and selecting "auto-connect shell sections" option. Both allow you to set the tolerance and thickness for the welds.

If the segments or any part are within the tolerance, this functionality will create a new part between them with the given thickness. New vertices might need to be created to make the connection, which should be centered at the area where the segments are the closest. If the thickness is given as 0.0, the connection will be set to a thickness equal to its length.

Translation of Shell and Solid Beam Sections

- This functionality can be accessed in the **HyperBeam** panel using the **section utils** radio button and selecting the **position shell/solid section** option. You have three options: center the beam section at the centroid, center the beam section at the shear center, or translate the beam section by amounts in the local Y and Z axes.
- The command for this is: `*beamsectiontranslate`

Export CSV Command Now Available for All Shell Beam Sections

In the HyperBeam GUI, click the **File** menu and select **Export CSV – all shell sections**. This will export CSV data for all shell sections in the model. In the previous version you had to select all the shell sections manually (and only shell sections) and use the right-click menu to export CSV data.

RADIOSS, Nastran, and OptiStruct Sections Now Display DIM1, DIM2, etc.

Dimensions for standard beam sections for the RADIOSS, Nastran, and OptiStruct beam sections are now shown as DIM1, DIM2, etc., and match the dimensions in the user's manual.

Component Names are Now Placed in Shell Beam Section Part Names

Part names created in the **HyperBeam** panel using lines, elements, or surfaces will now have the name of the component referencing those entities incorporated into their name.

Command File Support for the HyperBeam GUI

All changes to beam sections performed in the HyperBeam GUI will now be recorded in the command.cmf file. Running a command file where you made changes to beam sections should produce the same changes to your model. The commands for this are:

- `*beamsectionsetdataroot`
- `*beamsectionsetdatastandard`
- `*beamsectionsetdatasshell`
- `*beamsectionsetdatasolid`
- `*beamsectionsetdatageneric`

New Commands to Create Shell and Solid Sections

In the HyperBeam panel when the user creates a new shell or solid section, a new command will be called which more accurately reflects the options in the panel. The user experience will be the same. The new command will be compatible with running command files, which was not the case with the old command since it required a node ID be sent to it which the panel created internally. The new commands are:

- `*beamsectioncreateshelldirect`
- `*beamsectioncreatesoliddirect`

Resolved Issues

Saving single beamsection parameter edits issue is fixed.

Unused beam sections can be selected automatically in Delete panel:

- In the Delete panel if beamsects is selected as the entity, the preview unused button should be active. Pressing it should fill the mark with any beam sections that are not referenced by a property

All results are now calculated for all standard beam sections:

- Inside the HyperBeam GUI the results for beam sections are shown in the right hand window. In previous versions some of these results were missing or set to zero. In the current version all results are calculated for all standard beam section types for all solvers.

Back to top

11.0.120 Release Notes - Meshing

Map Mid-Mesh Thickness Utility

A new Map Mid-Mesh Thickness utility allows the automatic mapping of thickness from any legacy geometry to a representative FE model. There are several options to control how the thicknesses are applied to nodes, elements or properties plus auto-correction factors are available for complex use cases. A log file is available for a detailed description of how the thicknesses were mapped. Advanced visualization controls are available within the utility which enables quick evaluation of the thickness mapping.

Coarsen Mesh Utility

Added additional controls to the **Coarsen Mesh** utility, these controls include **Delete small parts**, **Component Size Factor** and **Auto Reduce Factor**. The latter two options enable a more detailed mesh output irrespective of the specified element size, it enables small complex parts to be captured correctly.

The performance and quality of output has improved significantly and the number of supported use cases has increased substantially.

Back to top

11.0.101 Release Notes - General

Model Check Tool

A new **Model Check** tool is now available. The **Model Check** is fully configurable therefore allowing control over which checks to include or exclude. The **Model Check** is controlled via a browser-driven user interface and the checks and results are categorized – error, warning and info. The information in the browser is also configurable; either an error view or HM entity based view is available. There is an auto and manual method to fix problems.

HyperMesh-ScriptView Debugging

HyperMesh-ScriptView debugging is now enabled via the **Scripting** toolbar.

Performance Improvements

Significant performance gains have been realized when exporting a solver file with includes over a network.

Midsurface Thickness Mapping Utility

- Significantly improved the performance
- Added support for second order elements
- Added new “Nodes, Properties on elements” and “Nodes, Properties on components” assignment methods for Abaqus

Back to top

11.0.101 Release Notes - API

New Mass Calculation APIs

- `hm_getcog`
- `hm_getmoi`
- `hm_getprincipalaxes`

ExtAPI Functions and Structures

New ExtAPI functions and structures to support component-based meshing.

Optimization Entity Support

Support has been added for all optimization entities for references/cross-referencing APIs, as well as for `hm_getentityvalue` and `hm_getentityarray`.

*createmark

`*createmark` enhanced with “by box”, “by cylinder” and “by sphere” primitive shape selection methods.

*equationhandle and *vectorhandle

Added the ability to turn off equation and vector entity handles via `*equationhandle` and `*vectorhandle`.

*geommatchtopology

`*geommatchtopology` has been added to modify geometry to better match its topology definition. This is to aid in accurate geometry export.

New Meshing APIs

Added new meshing APIs `*check_symmetric_mesh`, `*check_symmetric_surfaces` and `*map_symmetric_`mesh to create a symmetric mesh on symmetric geometry.

*compute_midmesh_thickness

`*compute_midmesh_thickness` has been added to calculate the thickness of a midmesh and assign properties.

Resolved Issues

Fixed crash with hwbrowsermanager view flush <state> API when no browser views existed.

Back to top

11.0.101 Release Notes - Connectors

Autopitch

Autopitch is introduced and is a tool which determines weld locations based on flanges and flat regions and will create connectors at a predefined pitch. It is found at ***Connectors*** > ***Create*** > **Autopitch** menu.

Feature Angle Recognition for Mesh Transition

Feature angles are preserved depending on the feature angle in the Options panel; features >25° are always kept.

Edge Snapping for Quad Seams

For all quad seams the connector snaps to close edges first before starting with the projection. This prevents small elements in case of imperfectly positioned seam connectors.

Seam Direction

The determination of the seam direction is much more reliable, especially for lap welds. It’s no longer necessary to reverse the direction as often.

Spot Quad Transition

Spot quad transition creates a 2x2 quad pattern around each spot on all layers. The orientation of this pattern has been improved, so that features and free edges are taken into account.

Imprint Pattern

When the imprint pattern is close to a feature or free edge, the pattern is by defaulted allowed to snap to such feature. This prevents small elements.

Sliver Elements

When the imprint pattern is close to a feature or free edge, the pattern is by defaulted allowed to snap to such feature. This prevents small elements.

Skip Imprint

A new option to create just the quad pattern, without doing the final imprint has been introduced. This allows you to manually resolve conflicting mesh situations.

New Seam and Spot Realizations have been Introduced

- seam-quad angled+capped+T,
- seam-quad angled+capped+L
- acm shell gap + coating
- pie rigid spider

Connector Imprint

The Connector Imprint functionality has mainly been introduced to provide the option to easily resolve conflicts between imprinted seams. You can create the imprint pattern, resolve the situation manually with all available tools in HM and then use the manual imprint functionality to imprint any shell elements into other shell elements.

The new mesh imprint capability allows you to use bolt connectors in a complete new way. During the bolt realization you can:

- close holes
- repunch holes
- reposition holes
- modify hole diameters

- create a certain number of nodes around the holes
- add washers
- fill holes with a certain pattern
- etc.

Bolt and Stack Panels

The right side of the **Bolt** panel has been structured to accommodate these new options. A **Stack** panel for hole detection and realization details has been introduced as well.

Post Script Handling

The current connector technology is not flexible enough to prescribe the exact organization and property assignment for the FE representations of a connection. This is especially true for all the realizations which have a postscript assigned the case. With the Post Script Handling update you can now skip the default postscript and define the destination component and property to be assigned.

Connector Options Page

A **Connector Options** page has been introduced. Currently only the field for the feconfig file location has been moved to that page, but others will follow. This is to make the panels more structured.

LLINK/PLINK PAM-CRASH Connections

The LLINK/PLINK PAM-CRASH connections normally don’t need a realization process, where we have to prove in HyperMesh which links have to be taken into account for the connection. This can be done by PAM-CRASH itself. We allow the connectors now to preserve more links than needed for a valid projection. All potential links are stored on the connector and consistently written into the LLINK/PLINK card.

Seam Connector Panel

In the **Seam Connector** panel, a **partition** subpanel has been added. This functionality proves whether each test point has valid projections in the given tolerance. The connectors are sheared into failed and passed pieces and organized as predefined.

Back to top

11.0.101 Release Notes - Geometry

Resolved Issues

- Continued robustness and accuracy improvements for surface and solid editing.
- Continued robustness and accuracy improvements for midsurface generation.

Back to top

11.0.101 Release Notes - HyperBeam

Shell and Solid Section Sketcher

Allows users to create, draw and update shell and solid sections in HyperBeam.

Interactive Section Editing

Allows users to edit section parameters and move vertices via click and drag.

View Controls (Zoom, Fit, Pan)

Added section view manipulators as icons and mouse control.

Reorient Shell Section

Added options to control the local origin, set the y-axis, and reflect shell sections.

Preview Unassigned Sections

Available in the HyperBeam panel under 'section utils'. This will store any unused beam sections in the user mark These options are available under the edit menu option for shell sections only.

Back to top

11.0.101 Release Notes - Meshing

Map as Circle

A new method called **Map as Circle** has been added to detect circular features, therefore meshing the features with a predefined mesh pattern. This mesh pattern can also be applied when in the secondary **Automesh** panel, either via **Autodecide** or **Map as Circle** mesh methods.

TetraMesh Optimization Utility

The new **Tetra Mesh Optimization** utility is located in the **Utility Menu**. It can take an existing tetra mesh and either move nodes or remesh to meet the required parameters. It has the ability to constrain trias, feature lines, nodes or elements within a refinement box.

Param File

Added new options for BatchMesher in param file under ***Other options*** > ***Edges equivalencing with tolerance:***

- Allow t connections
- Within components only
- Apply aggressive stitching

Added new option in the **Surface hole recognition** portion of the param file. The new option is called **Elems mode** and the option can be set to **minimal** or **exact**. This new option controls the number

of nodes around a hole and washer layer. In previous releases the nodes around a hole could not be rigidly enforced as user input.

Improved Logo Removal

Improved logo removal – “concave factor” added which enables more complex features to be detected and subsequently removed.

Performance Improvements

Improved handling of beads with flat plateaus and flanges.

The performance of the Tetra Mesh “mesh to file” option has improved significantly when generating large models in the order of 40 million elements or more.

OSSmooth Panel

Updated **OSSmooth** panel layout for improved usability. Also added **connection detect** feature which removes disjointed regions and ensures that the design material paths are bridged and connected together.

Back to top

Highlights of the HyperView 12.0 Release

- Symmetry Visualization
- 3D Stereoscopic View
- Section Cut with Path Plots
- Isosurface on Parts
- Measure Fonts and Colors
- General Enhancements
- Resolved Issues

Contents

11.0.130 Release Notes

Element Forces and Moments

Full Tensor Plots

New Visualization Option

User Defined Systems Definitions

Resolved Issues

11.0.120 Release Notes

Performance Improvements

Min/Max of Linear Superposition and Envelope

Video Overlay

Leader Lines

Image Capture Toolbar

Undeformed Geometry

Deformed Shape

Build Plots Panel

Advanced Query

Animated GIF Export

External Resources That are Added for Result Math Purposes

Resolved Issues

11.0.101 Release Notes

Build Plots Panel

Layer Filter

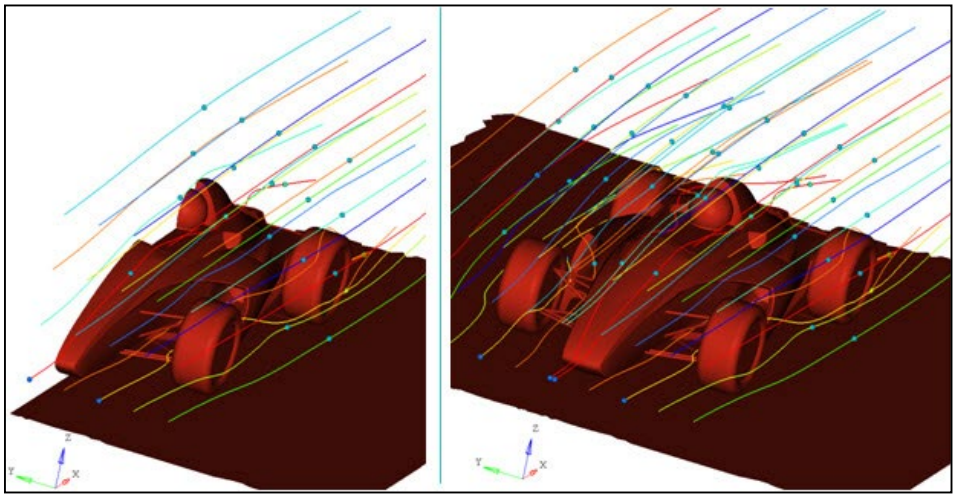
Framework/GUI

Image Capture Toolbar
Result Math
Result Browser
Averaging Elemental Results
Envelope
Layer Aggregation
Tracking Systems
Scalar Results
General
Resolved Issues

12.0 Release Notes

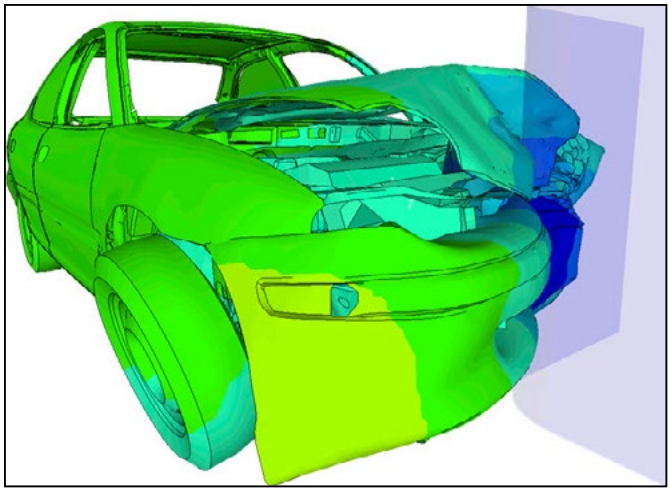
Symmetry Visualization

Symmetry visualization allows partially modeled geometry to be shown in full extent by reflection and/or duplication. Computational efficiencies are gained by modeling only partial geometry to the solver with appropriate boundary conditions, and visualization the full model in HyperView. Multiple planes of symmetry are supported. In conjunction with planar reflection, users can copy the geometry in rectangular and circular manner. Global directions or user specified local axis is supported. Result plots like Contour, Vectors, and Tensors are also shown on the reflected geometry. Symmetry settings are saved in the H3D files for visualizing the full geometry in HyperView Player.



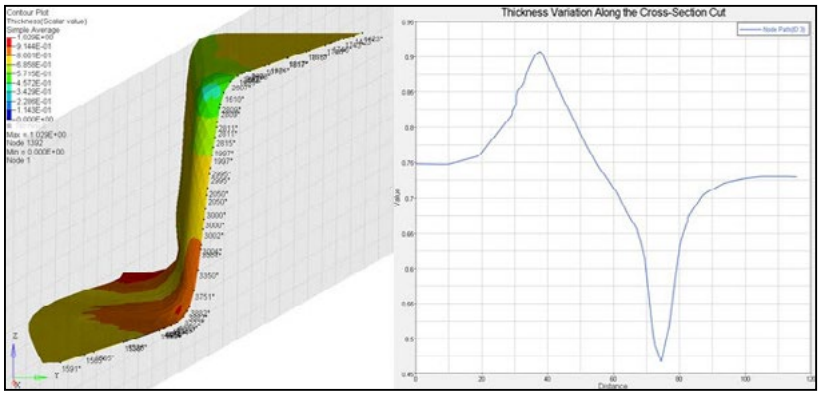
3D Stereoscopic View

HyperView supports stereoscopic 3D viewing to visualize simulation data in 3D. The visualization mode enables engineers to organize design reviews with other users and management in high end graphic visualization in a 3D environment using the Perspective view and depth perception. As a design review tool, the 3D stereoscopic views allow detailed analysis of the physical event being simulated. Modeling of spatial reality aids in enhanced interpretation of results within the familiar 3D environment.



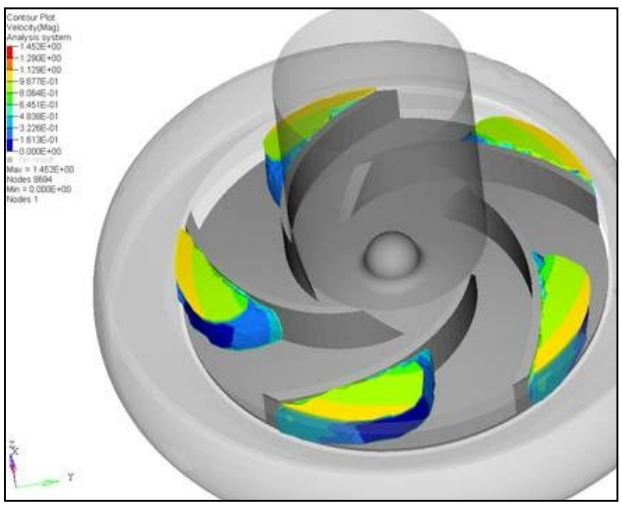
Section Cuts with Path Plots

Result variations along a section cut can also be studied by defining a node path along the cut. This helps engineers to assess the variation of a result along a user specified path on a cut section, as an example thickness variation in a stamping process. Measures can be defined on the nodes of a cross-section enabling a deeper analysis of the deformation of the section. Typical applications of measuring values on the cross section include plotting the position or distance between nodes on the cross-section, as measuring the intrusion.



Isosurface on Parts

Isosurface can be applied on selected parts, as an example on fluid elements, while visualization the structural components in full. For ditching simulation or fuel tank sloshing modeled with multiple fluids, the isosurface generates the fluid boundaries without influencing the structural components.



Measure Fonts and Colors

Font types and colors can be set for measure groups separately. The controls are similar to Notes with the flexibility to adjust the size, type, and color.

General Enhancements

- Assemblies are supported for the Selector, and Show/Hide/Isolate modes from the browser. This allows the users to review all the parts in the assembly by just selecting any part on the graphics area. Similar to the component review modes, holding the left mouse button down, the middle mouse scroll wheel can be used to depth zoom inside the model for selecting different assemblies.
- An entry field for notes allows inserting only the file name of the result file or the model file without the complete path of the directory structure.
- Publishing into PowerPoint is enhanced by allowing users to specify a master template or optionally control the resolution of the images. See the HyperGraph section for more details and the features supported.
- H3D files can be embedded in Powerpoint by a drag and drop operation, and viewed by HyperView Player in-place during slide show modes.
- The cross-section manipulator is updated to a 3D manipulator for X, Y, or Z-normal planes. The 3D manipulator can then be translated in any of the 3 directions to adjust the base point.
- Native dialogs on Windows Operating systems for File Open and Save show the Recent Places, Favorites, and the mapped network locations on the side for frequent access. Also, the drop down for file name entry field shows the recently accessed files.
- The information about a result math template used to load a particular model is shown in the Model Info tab (right click in the browser to select Model Statistics). It is particularly useful when loading from a session file to know what template has been specified for a given model.
- Improved graphics output during image capture and printing is implemented. Printing high quality images and captured images appropriately capture the graphical elements like notes, vectors, measure texts, legends and global triad, so that these elements scale up in resolution/sharpness along with the rest of the output.
- Ply names are sorted in alpha-numeric manner for convenience in the layer filter dialog.

Resolved Issues

- Captured images from the command layer adhere to the resolution specified in pixel dimensions.
- Powerpoints with embedded HVP objects are no longer restricted to the machines that created them. The embedded H3D file in the Powerpoint can be opened across user machines.
- The node path drawing is optimized to provide an improved frame rate when a lot of text is displayed on the path. This is more prominent with a node path covering a large number of nodes and text like id and values are displayed on the path.
- Application crash resolved when using the Entity attributes panel with Assembly Hierarchies in certain cases.
- Due to a regression introduced in an earlier patch, measures could not be created from the Advanced Query dialog. This issue is resolved.
- Improved performance in View synchronization operation between HyperMesh and HyperView windows that are coupled side-by-side.
- A significant improvement in changing the visualization options when dealing with large number of parts. Changing modes like wireframe to shaded or transparent with feature lines etc are much quicker from the Toolbar buttons or Entity Attributes panel.

- Deleting pages from that Session Browser that are not active caused a few refresh issues in the graphics area. The refresh issues are addressed.
- CaptureAnimation API has been updated to correctly handle special characters like “.” specified in the file path. A related issue resulted in truncation of filenames when capturing GIF files or other video formats from the Image Capture Toolbar and filenames have a “.” period. Both of these issues are resolved.
- Transparency settings from the Display Options is correctly applied when Accelerated Graphics is active.
- Multiple issues in FLD panel are addressed that involving Copy/Paste page, reading data from a file or selecting components on the screen with Copy/Paste.
- The templex expression for Live linked curves get updated to relevant measures when applying report templates in overlay modes.
- Deformation scaling is correctly captured and applied from report templates.
- Application crash issue resolved when reading a session file with invalid datatype names created as result math expression (for example a “.” period). A warning is posted instead.
- Result math plugin scripts give access to any optional arguments (strings) passed to the script. The plugin scripts can be Templex/Tcl/HyperMath, which accept optional arguments in addition to the datatype.
- Performance restored with automation scripts that don’t properly release stacks of “handles” used in the script.
- When creating an Iso plot and turning on the Color option, the color selected did not get applied to the entire set of parts visible. It only applied to a portion of the model that is “cut” by the Iso. This issue is resolved now.
- Preference statement to control the visibility of gridlines for section cuts is implemented to work correctly whenever new sections are created.
- Plot styles that are saved with datatypes names having special characters like &, % etc are restored correctly on import.
- RADIOSS Animation files exported to H3D with compression correctly captures the erosion state of the model.
- The model disappearance in perspective view when zoomed in too far is corrected by adjusting the clipping plane.
- Notes and Query updated to return all the components of the tensor (normal and shear) when queried.
- Precision of numeric calculations for principal values on a tensor is improved to reduce round off errors.
- The default text alignment in Notes is changed to left, unless overridden in the preference file by some other value. Model info note will always have the text alignment as “right”.
- HyperWorks Desktop window pops back on top while loading session file interfering with other applications. The issue is resolved so that the only time the main application window pops to the top is when an error is posted to the message log and the message log window is displayed.
- The Expression Builder for result math is updated to accept numbers with higher precision up to 12 digits after the decimal.
- Report templates with a group measure (single curve) restores the live linked curve correctly. The measure Templex expression wasn’t getting parsed correctly earlier.
- Performance improvement made to turning on/off the display of assemblies from the browser.
- Preference statement to set a user defined starting orientation of the model, *SetOrientation, is

supported to work correctly. Possible values are top, bottom, front, back, left, right, iso. Once the model is loaded, the view orientation will be set based on the value of this preference statement.

- Application hang resolved in some situations when a rapid dynamic zoom is performed in and out of the model. The mouse click processing is adjusted to sequence the operations
- The 3 point alignment for positioning 3D image planes is made robust for different formats of the video files.
- The display icons for components of an assembly is updated correctly in the browser after isolate operation on assemblies or parts.
- Note and Measures autohide state is correctly represented in the Image Capture.
- Model rendering code updated to handle certain cases of deformable cross section cuts not showing the elements correctly.
- H3D export updated to show right animations for a model that has tracking system applied on hidden parts at the time of export.
- Updating a cross section from planar to deformable should no longer affect the eroded elements display.
- Query panel export of csv or HTML throws up an error when a model input deck has empty parts in it. The application error is subdued in those situations.
- Video capture code is corrected for Mac to show the graphics area content during Capture dynamic rectangle video.
- Derived loadcase API is restricted from allowing recursive additions of same loadsteps any more.
- The compression factor set is Export Options for the H3D also gets applied to the H3D files of the Published HTML reports.
- Streamlines are captured in H3D generated from a Published HTML.
- The mesh visualization style is not applied until selected from the browser. Previously, just bringing up the pop up dialog changed the mesh visualization mode.
- Multi-select option is disabled for node path measures as the selection is order dependent.
- The browser interface is refreshed instantly to reflect up to date information of loadcase and timestep. There are many instances in the GUI that was not up to date with the active loadcase and loadstep, unless the browser view was manually refreshed or changed to a different view. These are the situations of stopping animation at random time steps, or changing a loadcase or loadstep using the keyboard shortcuts like “Insert”, “Delete”, “Home”, “End”, “Page Up” and “Page Down” buttons. The loadcase and loadstep changes are updated in the browser entry immediately. Also, when the animator’s current time is changed, the simstep entry in the browser (loadcase view of browser) is made bold. Conversely, when current simstep is changed using context menu in loadcase view, the dropdown entry for simulation is updated in the browser.
- Session loading restores correctly and without any errors derived loadcases defined across multiple result files.
- ApplyStyle is updated to include corner data setting for Tensor plots.
- Measure API GetValueList to get item list for node path measure returns only a single list now. Any scripts using this API would have got duplicate lists earlier.
- Model Info dialog provides more detailed information about element types and their count.
- Rotation speeds of the models in the graphics area is maintained unaffected by the animation speed set to lowest. Sluggish rotation resulted earlier when the animation slider was set to the lowest value.
- Copy/paste of HyperView windows maintains the undeformed model display settings.

- A session file loaded from a command prompt or double click in Windows explorer correctly loaded the model and result file, however, the session did not remain active prompting the user for a file name when saving. This issue is resolved.
- The Previous View behavior is made consistent between HyperView and HyperMesh
- A negative multiplier for contour is enabled.

Back to top

11.0.130 Release Notes

Element Forces and Moments

Element forces and Moments (from RADIOSS and Nastran solvers) are supported for shells. Forces and moments have different drawing modes for easy interpretation. The results can be resolved and transformed in different coordinate systems (user defined) and queried for export.

Full Tensor Plots

Tensor plots are enhanced to display all the six components on the tensor (normal and shear components). The shear components are proportionally drawn inside the elements being reviewed. Options provided to control the color of the tensor arrows in each direction and display the values at shear and normal components separately. With support of this feature, a full tensor can be queried in one instance exported to H3D or ASCII files (`csv`).

New Visualization Option

A new visualization option allows part colors to be displayed on entities that have no result when contouring. The default gray color for N/A result (or any user selected color) can be overridden with this option. By default the option is turned off (***Preferences*** > ***Options*** > ***Appearance***). While drawing in part colors with contours can produce interesting pictures for display. However, a frame rate reduction is seen with the option turned on.

User Defined Systems Definitions

User defined systems definitions are updated for Linear and Modal animation modes. The undeformed position (model location) of nodes is considered in defining the system. For Transient animation mode, the deformed position (current location at the time step) is considered. Once a coordinate system is created, the orientation remains fixed independent of how the model deforms.

Resolved Issues

- A critical issue identified in 11.0.120 that resulted in spurious hangs when reading model/results on certain Windows machines has been resolved. The issues were limited to reading Nastran (Op2) and LS-DYNA (d3plot) along with any other combination of the solver decks, although it isn’t reproducible every time. However, a fix is included in the current patch that resolves the application hang in all these scenarios.
- Published files (as HTML or PPT) on Linux machines had limited access permission requiring only the owners to delete the files. This restriction is removed, allowing all authorized users in a group to delete if required.
- Hidden notes from the background were shown in the images captured.
- When replacing only a result file in the active window, the model note still showed the old file name.

- State of the contours applied on selected assemblies are saved and restored correctly in session and report templates.
- Contours can now be applied on parent assemblies via the context menus in the browser.
- If some of the nodes on solid elements did not have results, the iso surface wasn't plotted correctly.
- An issue has been resolved where the animation time step for models loaded in HyperView was getting reset to 0.0 when switching between HM and HV clients.
- Values measured on contour and iso-surfaces plots are now consistent, particularly with mid side nodal values for 2nd order elements.
- Menus got duplicated when a session file (mvw) is dragged and dropped on application shortcuts. This issue no longer occurs.
- Window resizing issues when HM is present in the Desktop is resolved.
- When doing a contour plot and then using the Transparent option either in the Section Cut or Iso panel, the contour is turned to grey on various portions of the model depending on the current angle at which the model is viewed. Same happens without a section cut or an iso surface, if some components have no result and are using T or Boundary Geometry > Transparent. The graphic artifact is resolved.
- Live linked measures on clearing contours and reapplying for large models resulted in memory leak on Linux machines.
- Dot operator now returns a scalar as expected. This was broken in 11.0.120 but is fixed in this release.
- H3D export of contoured results with some RADIOSS models did not bring back the same contours because of conflicting duplicate IDs in the models.
- HwAscii reader has been updated to correctly read in (complex) stress tensors from an FR analysis containing multiple loadcases.
- Performance fix has been implemented that addresses the slowness in rotation of large models containing masked hexa and penta elements.
- User defined systems, by node option, considers the nodal location in undeformed position for Linear and Modal animation modes. This is a change in behavior compared to previous versions. For Transient mode of animation, the deformed position (current location at the time step) is considered when creating a system. Once the system is created, the orientation remains fixed independent of the model deformation.
- An H3D model embedded in a Word or PowerPoint document by drag and drop method or by publishing did not refresh the image. Users needed to double click the image to see the preview. A change in image position for 11.0 resulted in the issue resolution.
- A folder named Altair_dab_10 would be created in the launch directory when running HyperView Player. This no longer happens.
- Quick query in HyperView Player did not work in slide show mode for PowerPoint 2007 (when HyperView Player exits).
- Plot styles applied for tensor plots correctly captured the definition but did not apply the settings when used. This issue is resolved; all tensor plot styles apply the saved settings as intended.

Back to top

11.0.120 Release Notes

Performance Improvements

Significant performance improvements for importing the Abaqus, LS-DYNA, Nastran and OptiStruct input decks in HyperMesh and HyperView when importing from a network drive. The percentage gain varies depending upon the network.

The network performance improvements are also extended to importing, contouring, and animating Nastran Op2 and LS-DYNA d3plot files in HyperView. The percentage gain varies depending upon the network (2x to 10x improvement).

Dramatic improvements in animation speed and model view manipulation (rotate/pan) is available taking advantage of the hardware. A graphics performance improvement of 2x to 6x is possible with a moderate graphics card, and the gains are amplified with high end hardware. Updated drivers may be needed to realize the effect. ATI cards are not supported at this time.

Loading and animation of stress contour results for flexible bodies have been sped up by a factor of 4x, compared to version 10.0 (to realize the maximum gain in performance, solver patch 11.0.202 is required to generate the H3D files containing the flexible bodies).

Min/Max of Linear Superposition and Envelope

A second level of derived loadcases for finding the min/max/extreme of multiple Linear Superposition and/or other Envelopes is supported. This feature allows performing multiple linear combinations or envelope results in HyperView and extracting the overall min/max/extreme of these combinations. The definitions of derived loadcases can be saved in the session files and report templates for reuse on different iterations.

Video Overlay

The 3D Image Planes that are embedded in model space can be incrementally adjusted to match the model dimensions. In addition to using the 2-point and 3-point alignment for scaling the image planes to the model size, the image planes now allow minor adjustments to the rectangular plane by the sizing manipulators available on the periphery. This helps in making small size adjustments without necessarily going through picking alignment and scaling points again. Small changes to plane orientations can of course be adjusted by the tripod manipulator attached to the image planes.

Leader Lines

Thickness of the leader lines for notes can be adjusted by controlling the thickness of the note border.

Image Capture Toolbar

The Image Capture toolbar remembers the previous file type selected. If in a session the file type is changed to JPEG or TIF, subsequent launch of the file save dialog will default to the previously set type.

Undeformed Geometry

Undeformed geometry is now affected by the section cuts. This is helpful in seeing the cross sections before and after loading, a comparison of the deformed and undeformed structure. The undeformed geometry display state and its properties are set from the options in the **Deformed** panel.

Deformed Shape

Deformed shape of the model can be exported in STL format (From the Standard toolbar, or by clicking ***File*** > ***Export*** > ***Solver Deck***).

Build Plots Panel

An extended dialog for sorting and filtering results is available in the **Build Plots** panel.

The **Build Plots** panel now allows plotting of part bound results. The part bound results are typically available from math expressions defined in an XML file. The same XML file can be loaded in HyperGraph, with the advantage that of Build Plots in HyperView, parts can selected graphically on the screen.

Advanced Query

Part bound results are also supported in Advanced Query. Using result math to create expressions for part summations or max/min, the Advanced Query can be looped through the entire model and all loadcases to find out the Top N/Bottom N ranked parts.

Animated GIF Export

Animated GIF export has more flexibility in controlling the playback speed and size of the GIF image. All the animated GIFs exported in video capture or publish session can take time delay between the frames and a size specification of 100%, 75%, 50%, or 33% (Preferences > Export Settings > GIF).

External Resources That are Added for Result Math Purposes

External resources that are added for result math purposes (in the Result Math Expression Builder, Add Resources) are available as file parameters for report templates. Any constants file or additional result files added as a resource in baseline iteration can be interchanged or updated when applying a report template for subsequent iterations.

Resolved Issues

- Memory usage on animated GIF export is reduced in half. This is significant when exporting a large number of timesteps/frames on low end machines with limited RAM or 32-bit machines that have process limits set by the OS. If the rate of memory consumption is too high compared to the actual physical memory, a warning is posted to the user that the process is likely to fail, and offer a choice to abort or continue.
- Specular highlighting is corrected so that there are no inconsistencies in the display of the model. In earlier version of the software, some settings demonstrate few areas of the models displayed properly highlighted and others not.
- Model info note updates to correct file names if a result file is changed while reloading with the same model file in any HyperView window.
- Using ApplyStyle between models that have SETs defined in the solver input deck, crash would result when one of the source or target models do not have SETs defined in the solver input deck.
- Linux machines that have certain wide spaced fonts installed would cause HyperView to crash when changing the font types in the Font selector. As the fonts installed on a Linux machine vary from distribution to distribution, user preferences, etc, application crash is prevented whenever a problematic font is encountered in rendering.
- The extent of model rotations using the side and up/down keys is controlled by the angle specified under user preferences (Preferences > Options > Visualization > Rotation Angle).

- Loadcase tab in the browser would show up blank when switching from HyperGraph client to HyperView in some instances.
- The circle zoom drawing in the graphics area appears correctly at the mouse location when View Synchronization is turned ON.
- TensorLocalToGlobal operator correctly returns a complex tensor in global system - used be in Elemental system previously, which is incorrect.

Back to top

11.0.101 Release Notes

Build Plots Panel

- A simplified Build Plots panel is available in HyperView to directly plot data into HyperGraph from the animation window.
- Results associated to nodes/elements/parts/systems can be plotted by graphically selecting the entities providing visual feedback.
- XY plotting of transient and modal (complex) data is supported.

Layer Filter

- Filter results by ply orientation or material by using Layer Filters. User defined filters enable processing results across specific plies.
- A layer filter reduces the layer list available for aggregation in the Contour, Vector, Tensor, and Iso panels for the data types that have layers (for example, plies in composite stress/stains). This is important in composites post-processing where different set of layers (plies) belong to different materials, or if different set of layers are oriented in certain directions (0/45/90 degree plies), and it is of interest to find out the maximum stress or minimum failure index in all the plies oriented in a certain direction or that are made of particular material.
- Whenever a data type with layers is selected, the filter is enabled (orange highlighting around the icon shows that the filter is turned on): Once the layer list is filtered, the aggregation of Min/Max/Extreme/Sum/Average/Range can be performed on the reduced list. The layer filter is available for aggregation in the Contour/Vector/Tensor/Iso panel for the datatypes that have layers (for example plies in Composite Stress/Strains).

Framework/GUI

- Rename and Delete report layers have been enhanced.
- Animated GIF supported for Publish session is now available.

Image Capture Toolbar

The **Standard Image Capture** and BMP Image Capture toolbars are consolidated into single toolbar called **Image Capture** toolbar that is available on all platforms. On Linux, the **Save Image to File/ Clipboard** option can only be set to “Save Image to File” and “Capture Dynamic Rectangle” and “Capture Frame Area” functionalities are not available.

Result Math

In HyperMath, new operators were added – Principal Axis Vector, TensorFromScalar, and VectorFromScalar

Result Browser

- All the images in the **Result Browser** are PNG.
- The Assembly folder in the **Result Browser** now behaves exactly as it does in HyperMesh browsers (the assembly itself can be turned on/off and it turns on/off all children).

Averaging Elemental Results

New options for averaging of elemental results, Maximum and Minimum, are added. The averaging method converts element results to nodal results by Max/Min of surrounding elements. Assign the hottest or coldest value from the neighboring elements without smearing the absolute values.

Envelope

Range. Obtain the spread of results across all loadcases.

Layer Aggregation

Layer aggregation now includes Sum, Average, and Range.

Tracking Systems

For tracking systems defined by a Plane, the local axis of the tracking systems can be aligned with any specific global direction by specifying the plane type. Available orientation options are OXY, OYZ, and OZX. This flexibility is useful in situations where the global coordinate system alignment differs from the model coordinate system.

Scalar Results

Resolved-in system option within the **Contour** and **Iso** panels is disabled for scalar results. The scalar results by nature are unaffected by any coordinate transformation.

General

A new environment variable HWPOST_REMEMBER_LASTREAD allows the model and result file names to be remembered for subsequent application launches. Default is to remember between sessions.

Resolved Issues

- **Framework**
  - Removed additional unnecessary redraws for certain operations.
  - We now support the Templex, Reports and Animation panels in all clients (including HyperMesh). The ESC key can be used to dismiss them and take the user back to the previous client panel. This includes enabling the Reports toolbar for HyperMesh client and enabling the Templex button on the Scripting toolbar for the HyperMesh client.
  - Tab behavior is improved when switching clients, using left and right tab areas, and turning tab areas on/off.
  - Refresh issues of the graphics area with copy and paste window or copy active model and paste active model resolved.

  - Command line start up errors when loading session files initializing the toolbar with interchangeable clients are now resolved.
  - Having any of the Show/Hide/Isolate/Selector modes and selecting multiple components on the screen, new session crashed the application. This has been fixed.
- **Model Browser:** Autofit configuration options in the Model Browser now works with Show/Hide/Isolate options.
- **Result Math**
  - In HyperMath, the binary operators add and subtract (+, -) now correctly ignore N/A results.
  - Restoring session or report template files containing image planes with duplicate names resulted in a crash has been resolved.
- **Video Overlay:** While using video overlay, image planes orientated by 3-point alignment has been corrected in few cases.
- **Miscellaneous**
  - Graphical issue of displaying duplicate solids faces is resolved, no voids are shown.
  - Publish options set on a page level of the Session Browser are retained, and trickled to each of the windows in the page.
  - Advanced query enabled for querying corner bound elemental results – a regression fixed.
  - H3D Export of Linear Superposition results/strain tensor and complex has been corrected.
  - HvTrans performance improved while handling large HwAscii data files.
  - Application crash with report templates bringing in models with empty components has been resolved.
  - Several application crashes resulting from active model cut/copy/paste have been resolved.

Back to top

Highlights of the HyperGraph 12.0 Release

- PowerPoint Beta

Contents

11.0.130 Release Notes

HG Enabling Units

Multiple File Selection

Resolved Issues

11.0.120 Release Notes

Curve Preview

Metadata Table

ADAMS .res

Bessel Filter

Resolved Issues

11.0.101 Release Notes

HG Trans GUI

IIHS Intrusion Templates

Python Function Interface

Context Menu for HG3D

Diadem XML

Build Plots Filter

dB10/dB20 Reference Value

Range-pair Function

Define Curve and Coordinate Info Panel Performance

Renaming and Deleting Report Layers

Resolved Issues

12.0 Release Notes

PowerPoint Beta

The HG PowerPoint Beta functionality is the commercialization of the MVP toolbar's Export functionality. This tool is an improvement to our current PowerPoint publishing capabilities, and was generated by our AEs in the field. It is a great example of how customizable our HWD products are. The PowerPoint Beta feature provides more options when generating Microsoft PowerPoint files. Engineers spend a lot of time generating these PowerPoint files for their managers and customers. This enhanced feature will make generating these reports much easier, decreasing the work of the engineer and allowing more time for engineering. This is done by providing users more control over the way the PPT will look before publishing.

What to expect when Publishing from each client (HM/MV/HV/HG/TextView/MediaView):

- HyperMesh – Image only
- MotionView – Image only
- HyperView – Image or H3D
- HyperGraph – image only
- TextView – Editable Text box (if your text expands outside the window, it will extend outside the text box in PPT.)
- MediaView – Image only
- HG3D – Not yet supported

Features:

- Title PowerPoint Beta
- Support of PowerPoint Master Templates
- Support of Title Attributes and Position
- Support of Image Positioning
- Support of Picture Resolution
- Support of Page/Window Export Selection
- Support of one window per PPT page
- Support of Font Settings for HyperGraph & HyperView objects
- Support of H3D Export Options

Back to top

11.0.130 Release Notes

HG Enabling Units

The HG Enabling Units is a powerful feature that makes working with units from results much easier. This feature is designed to work with 3 main usecases.

- Automatic Unit Recognition
  - RADIOSS abf files

  - Adams req(withnam file)
  - Xml files
- Setting Unit Profile
  - LS-DYNA Consistent unit settings
- Setting File Units
  - Abaqus, RADIOSS (Bulk Data) pch and OP2, Nastran pch and OP2

This feature is turned off by default and can be turned on via a check box in the ***HG*** > ***Tool*** > ***Options Panel***. When this feature is enabled it is set internally within preference settings at 3 levels (File/ Reader/Solver) that do not have to be set again. To turn these settings off or reset them go to ***Tools*** > ***Reset Units Association*** option.

Multiple File Selection

The **Multiple File Selection** dialog allows you to select multiple files with intersecting data (for example, multiple runs of a particular test) and plot the data from all files simultaneously in the current HyperGraph session. To access the **Multiple File Selection** dialog, click the ***Multiple File Selection*** button on the **Build Plots** panel.

Intersecting data refers to files that contain the same type, request and component information. If the selected files do not have data that intersects with the first selected file, the Type, Request and Component fields will be empty when you return to the **Build Plots** panel. After making your selections and closing the dialog, you are returned to the **Build Plots** panel where you can select the type, request and component options for the X and Y data. Click Apply on the **Build Plots** panel to plot the data for the selected files. From the **Multiple File Selection** dialog, you can define the curve names, curve color and line style, as well as generate file sets for the current session.

Resolved Issues

- Permanent preference file settings were returned to 10.0 status.
- Fit Range correction and improvement; and Auto Fit correction and improvement for existing Axis issues have been applied.
- Adams res reader was fixed to remove an issue
- SPMM txt file reader was fixed to remove an issue.

Back to top

11.0.120 Release Notes

Curve Preview

The HG Curve Preview allows you to quickly review curves before plotting them in the session. This function is designed to work with a single file or multiple files. It is turned on via a check box in the Build Plots panel and can be turned off by any of the following means:

- Unchecking the check box
- Leaving the panel
- Clicking the apply button
- When a different Y type is selected with different requests and components

Metadata Table

The HG Metadata Table feature provides an informational table of metadata that is read from the file. This will enable you to access a file metadata and paste it in Templex Statement supported entities (notes, axes, page title, window header/footer, etc.) for use in the HWD session. This information will be accessible through the following two interfaces:

- File > Load > Metadata table
- Icon > Load Metadata table

ADAMS .res

Performance increase while reading.

Bessel Filter

Analog Bessel filters are useful when there is a need to minimize phase distortion because they have a maximally linear phase response over the pass band. This means that all frequencies in the pass band will have the same time delay through the filter, thus tending to preserve the shape of a waveform.

Digital Bessel filters are of less interest because they lose this property as the cutoff frequency increases relative to the sampling frequency. The tradeoff for reducing phase distortion is allowing more amplitude distortion compared to other filter types.

Resolved Issues

- **Export Output Text:** Added back lost functionality when removing MainWin to export evaluated Templex data.
- **Templex:** Is working batch.
- **Vel curves from H3D file switched when reading session:** Velocity plots saved in session are correct now.

Back to top

11.0.101 Release Notes

HG Trans GUI

HGTrans is a result translator for HyperGraph that translates solver result files to ABF files. Users currently have the ability to run HgTrans in batch mode. This GUI will allow the users to translate the files using a much easier graphical interface and see what exactly they are translating. This GUI is designed to work with all supported HG readers internal functions and registered functions.

IIHS Intrusion Templates

In the Vehicle safety Tools preferences, we have now added two templates to plot IIHS front intrusion and IIHS Side structure intrusion plots for RADIOSS and LS-DYNA.

IIHS Front Intrusion works on:

- T01 file of RADIOSS

- nodout, binout and d3plot files of LS-DYNA
- HyperView h3d files

IIHS Side Impact Intrusion works on:

- A0 files of RADIOSS
- d3plot files of LS-DYNA
- HyperView h3d files

Python Function Interface

Added context menu support for HG3D just like the HG2D XY Plot.

Context Menu for HG3D

The **Standard Image Capture** and **BMP Image Capture** toolbars are consolidated into single toolbar called **Image Capture** toolbar that is available on all platforms. On Linux, the **Save Image to File/ Clipboard** option can only be set to "Save Image to File" and "Capture Dynamic Rectangle" and "Capture Frame Area" functionalities are not available.

Diadem XML

Support of new Diadem XML Format (`.tdm` and `.tdx` files).

Build Plots Filter

New filters for all plot types in the **Build Plot** panel.

---

dB10/dB20 Reference Value

Add reference value and weighting support to dB10/dB20 scale.

Range-pair Function

Added Range-pair function.

Define Curve and Coordinate Info Panel Performance

Improved data querying using the Define Curve and Coordinate Info panels from 11.0.

Renaming and Deleting Report Layers

This feature allows users to rename and delete report layers by right clicking on the layer name.

Resolved Issues

- Cross plotting animation files: Fixed the bug with cross plotting Data Types with animation files.
- **Axis Auto Fit** has been corrected. To fix an axis, uncheck **Auto Fit**.
- Circle zoom has been fixed when working in the **Coordinate Info** panel.

Back to top

Highlights of the MotionView 12.0 Release

- New GUI Architecture
- Project Browser Enhancements
- Integration with HW Desktop Collaboration Tools
- Modeling via Distributed Files
- Entity Panels and Toolbar Enhancements
- General State Equations
- MotionView Now Available on Macintosh

Contents

12.0 Updates

Project Browser Enhancements

Entity Panels and Toolbar Enhancements

Selection Mechanism Changes

General State Equations

Messaging Improvement

Changes to MotionSolve XML

Resolved Issues

11.0.130 Release Notes

Integration with HWDesktop Collaboration Tools

SLINCH Support

Box Graphics

Resolved Issues

11.0.130 Release Notes - Motion Vehicle Library

TNO Magic Formula and Swift Tires Support

Truck Tire System Definitions Enhancements

Single Tire System Definitions

Vehicle Models in Online Help

Resolved Issues

11.0.120 Release Notes

Modeling via Distributed Files

Systems/Analysis Toolbar

SpringDamper Panel Enhancements
Performance Improvements
Resolved Issues

11.0.101 Release Notes

New Panels
Model Browser (Project Browser)
New HyperWorks Desktop Framework
Session Browser
Context Sensitive Menu for Graphics Area
Delete Feature
Improved File Path Management
General Constraints Support
Additional H3D Output Options Supported
Entity Definitions Improvements
Saving of Properties
Improved Contact Modeling
Graphical Picking Enhancements
File Management
Load and Save Toolbar Panels
MDL Library Updates
Resolved Issues

12.0 Release Notes

New GUI Architecture

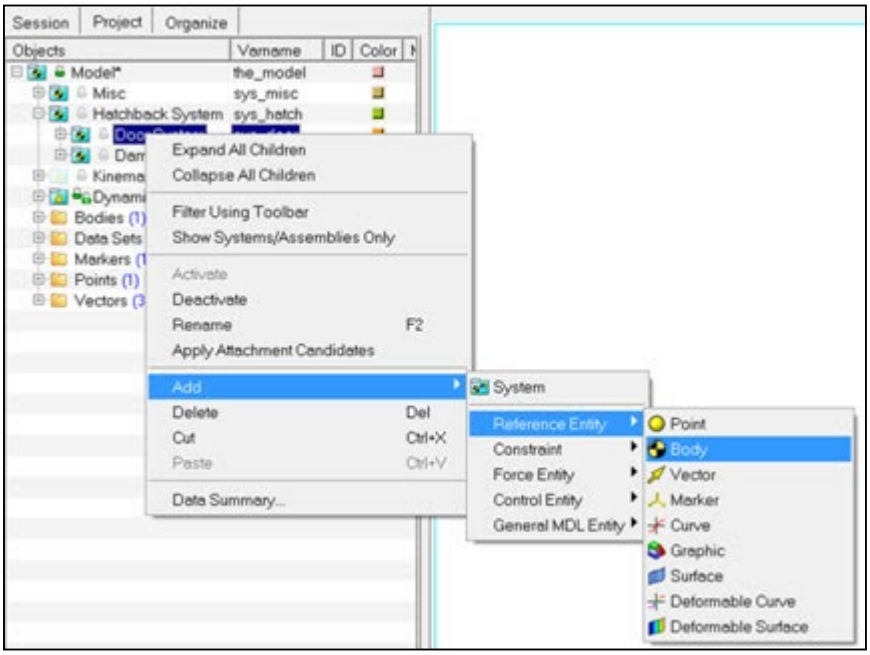
MotionView 12.0 comes with an enhanced User interface based on the Python programming language. The MotionView Project Browser, Entity 'Add' dialogs and panels are affected, though the layout, functionalities and user feel remain the same as in version 11.0. With this change, a significant improvement in usability and performance is expected.

Project Browser Enhancements

The new Project Browser comes with following enhanced features:

- Navigation using keyboard arrows and keyboard shortcuts to rename and delete entities.
- Search feature - direct type-in to find entities or use the Find tool
- The browser has an additional column that displays variable name of the entity.

- The content hierarchy is displayed collapsed to the first level when a model is opened
- Folders are now listed alphabetically
- Drop-down menu “View All Types” and “View Selected type” no longer exist. Filtering options are now available via right-click context menu. Entities can be filtered by entities based on entity toolbar selection (“Filter using Toolbar”) or specific entities (e.g. “Show Bodies Only”). Filtering also supports multiple entities, e.g. “Show Joints and Points only”.
- Improved context menu layout
- Cut’ now works with multi-selections
- Analyses will be always listed after all Systems/Assemblies
- Adding entities through context menu in the browser has an extra level

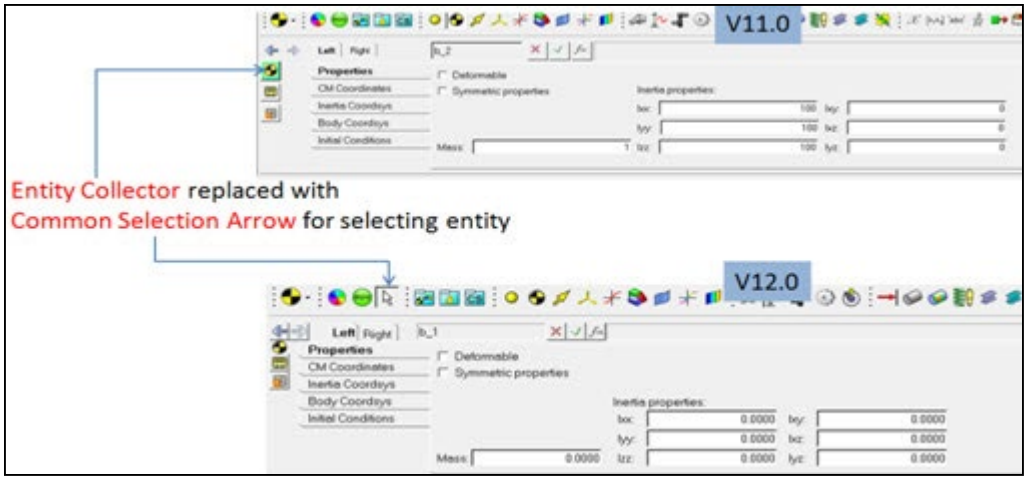


For more information on the **Project Browser**, refer to Project Browser in the MotionView User Guide.

Entity Panels and Toolbar Enhancements

In previous releases of MotionView, all of the panels in the interface had been constructed using TCL. With the 12.0 release, we have created a number of Python-based panels that are equivalent to, and replacements for, many of the TCL panels. Using the default Python panels, you should experience improved panel behavior while still maintaining the same overall ‘look-and-feel’ that you are accustomed to. Some of the panel improvements include:

- Less flickering when switching between panels
- Faster display of panels when navigating the model through the Project Browser
- More panel-to-panel consistency (e.g. text labels, widget placement)
- Wider fields for entering Expressions
- New logic for some GUI widgets, such as disabled checkboxes, that now replace some alert boxes
- Additionally, many of the entity Add dialogs have also been rebuilt using Python. Add dialogs come with keyboard shortcuts on OK, Apply and Cancel buttons.
- The entity collector in the panel is now replaced with a common entity selection arrow in the toolbar. The entity collector in the panel is changed to an icon image with tip text.



Selection Mechanism Changes

The mechanism for selecting model entities from the graphic screen has been updated. The MotionView toolbar now has a new **Entity Selector** button to indicate when you are in Selection mode. This button appears depressed when graphical entity selection is allowed and un-depressed when you have entered Collection mode. For example, when using a Body Collector on the Marker panel you are in Collection mode, so the **Entity Selector** button will appear un-depressed. To exit Collection mode, simply choose an icon in the toolbar via left-click and you are back in Selection mode for the entity type represented by the icon you have chosen. Or, you can click the Entity Selector icon.

Tip: Double-clicking the **Entity Selector** icon is a handy way to clear out any entity icon selection in the toolbar; it can also take you out of any active Collection mode.

Another related change is in the behavior of panels while selecting the icons in the toolbar. The toolbar now exclusively acts as a filter for selecting entities from the graphic screen. Mere depression of an entity does not bring up the panel for the entity of that type that might have been selected previously. For example, if a body was selected with its panel displayed and then a different entity was selected, clicking on the Body icon again in the toolbar does not display the previously displayed body panel.

Tip: Use the Back/Forward button on the panel to navigate to the previously selected entity panels.

General State Equation Entity

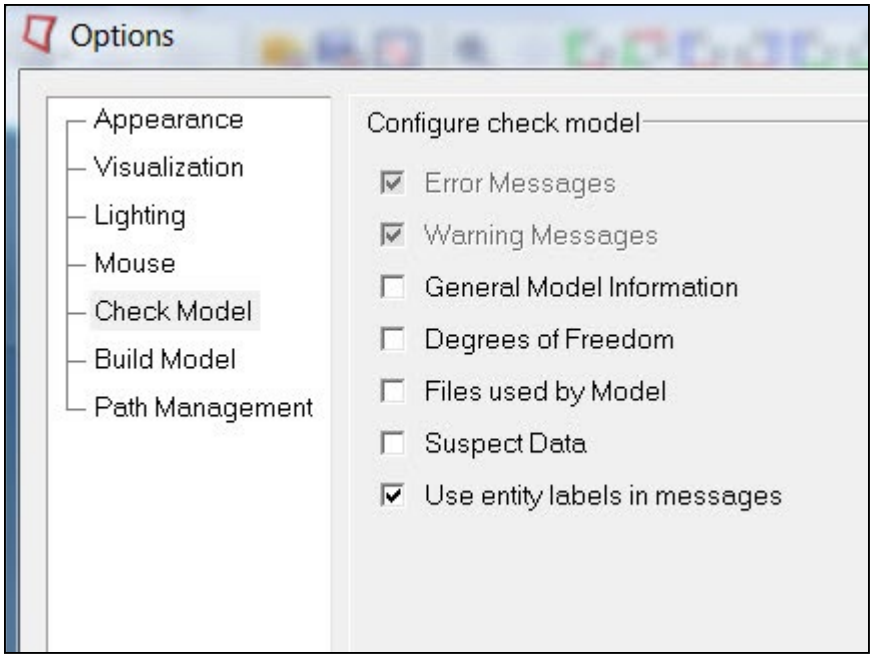
General State Equation (Control_StateEqn in MotionSolve and GSE in Adams) used to model a non-linear dynamic system is now supported in MotionView MDL through new statement *GeneralStateEquation() .This entity is not yet supported in the GUI. Refer to MotionView MDL Reference Guide for more details on this entity.

Messaging Improvement

MotionView message log now provides messages referring to entity labels instead of variable names. Shown below is an example.

MV 11.0	MV 12.0
Error: Unknown symbol BLANK Error resolving entity MODEL.b_0.cg.	Error: Unknown symbol BLANK Error resolving entity Link-cg under Model.

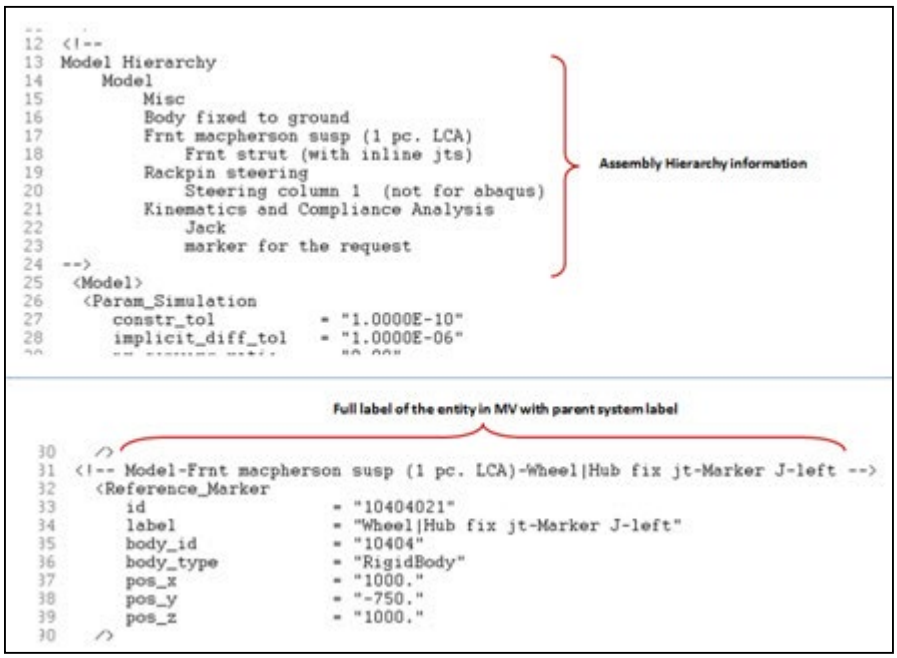
This option can be controlled in ***Tools*** > ***Options*** > ***Check Model***.



Changes to MotionSolve XML

From this release, MotionSolve XML exported by MotionView will contain additional modeling information such as Model assembly hierarchy and full label for each entity. The assembly hierarchy information is available at the initial portion of the XML before the `<Model>` statement.

The full label for an entity is available above each entity statement.



MotionView Now Available on Macintosh

From this release, MotionView is available in Macintosh platforms along with other Hyperworks Desktop products. In this version, MotionView interface with Collaboration Tools is not supported.

Resolved Issues

- Adding entities such as solver array and solver strings into a system use to result in those entities being added under Model.
- Entries in dataset were not being refreshed dynamically. This issue was noticed when one entry in dataset is dependent on the state or value of another.
- An issue in Contact Properties macro where pressing the ENTER key after an entry of a value did not register the change in value is fixed.
- Expression Builder falsely gave error on setting value as an entity ID inside a system.
- Using “Interactive Mesh” during CAD import to launch HyperMesh would not bring up the Utility Browser
- Slow performance while selecting multiple entities in the browser on a large model
- Inertia and Body coordinate frames were displayed when the Inertia CS and Body CS checkbox were turned off in the Body panel
- Resolving Filename datamember under a 2nd level child system gave application error
- Check Model skipped checking Unresolved state of Curve collector in Force and Motion panel
- Check Model fails to detect error in evaluating expressions in Solver Array panel
- Importing RADIOSS (Bulk) using CAD Import was not calculating mass of the bodies
- Some of the demo models in the install contained errors

Back to top

11.0.130 Release Notes

Integration with HWDesktop Collaboration Tools

This version brings in a first level integration of MotionView with HWDesktop Collaboration Tools a tool to organize, manage CAE data and collaborate in teams. This first release comes with the ability to create a MotionView specific library and to support MotionView related workflow within the collaborative environment. To learn more about available features and limitations refer to MotionView Interfacing with Collaboration Tools.

SLINCH Support

The FlexPrep utility for flexible body generation now supports specifying mass units “SLINCH” in bulk data file. SLINCH is equal to 12 slugs and is used in FE modeling as a derived unit where the force unit is pound force (lbf) and length unit is inches. Such FE models can be directly used to generate flex bodies without the need to scale the model to other mass units.

For more information on different types of units that are supported, refer to RADIOSS Flexbody Generation in MotionView User’s Guide.

The standard dataset and form to specify the modeling unit is also enhanced to support “SLINCH.”

Box Graphics

Creating box graphics now includes a default value for length.

Resolved Issues

- Pre-load values in the "slave" tab (either of left or right tab that is non-editable) of a symmetric torsion spring pair definition were being set to 0.000 has been fixed.
- Application error on Measure tab in Points panel with properties "Error Evaluating" is fixed.
- A critical browser issue that causes the MotionView Project browser to blank out and become non-usable while opening a session file (mvw) saved with an active HyperGraph client is now resolved.
- MotionSolve export of Action Only Force with explicit Markers will be written as <Force_Vector_ TwoBody/> instead of <Force_Vector_OneBody/> statement. When two Action only forces were defined using same set of explicit markers, the force output request returned the summation of the two forces. It was not possible to obtain outputs for the individual forces.
- The issue of writing extra markers into MotionSolve solver XML for output requests of type Expression or User Defined has been resolved.
- An application error while adding another attachment to a system or assembly when the label of an existing attachment is on edit mode is fixed.
- Application error while selecting local dll file in user defined tabs of Force panel is fixed. The error occurred while the operation was repeated after selecting a file in Run panel.
- Invoking the MotionView Panel online help by using the F1 key was not working. This has been fixed. Using F1 key on the keyboard opens the relevant panel related help page in MotionView Users Guide.
- CAD Import: Graphics and CG locations were not scaled properly when length units in MotionView were different from the input file. This issue has been fixed.
- CAD Import: Support for importing UniGraphics and Parasolid formats with mass and inertia has been restored. Solidworks & JT format is now supported.
- The issue of changing coordinate values of existing points upon CAD/FE import has been fixed. This occurred due to improper assignment of variable names to the point entities that were created upon import.
- Reversed signs for Product of Inertia values on bodies during import of CAD files is fixed.
- Improper size display of torsion spring implicit graphics while opening a model in MotionView with an existing model is now fixed.
- Drop down entity collector list in the Output panel is displayed at far top left of MotionView application window. This issue is fixed.
- A memory related crash of MotionView on models with large H3d files in Win32 is fixed. MotionView now provides message in case of insufficient memory during export.
- A critical crash of MotionView when expanding all children in the browser for a large model is fixed.
- Graphic colors of entities being changed permanently upon using the Find connected entities macro has been corrected.
- MotionSolve export of reference markers for body velocity initial conditions (vm_id & wm_id) specified in the Bodies panel, Intial Conditions tab is now supported. Issue of these marker information not saving to the model file (.mdl) is also resolved.
- Export of system or analysis definition did not contain default values for attachments. This issue resulted in unresolved attachments when a system or analysis was re-imported after export. This issue has been resolved.
- The "Locate" functionality on the Body Deformable panel that is used to position the flexible body is now working.
- There was an inconsistency on the time unit between the flex body h3d and MotionView. This caused erroneous scaling of frequency and default damping values when the time unit used to generate the flexible body was different than "second". This issue has been resolved.

- A critical crash while selecting a different curve from the browser when the field in Attributes tab of currently selected curve was active is fixed.
- Implicit graphics for some of entities, when selected, were not being highlighted in the graphic screen. This has been resolved.
- Double-clicking on selected entities in the model tree was not confirming the selection and closing the tree. This issue has been resolved. Double clicking on an entity in the model tree is equivalent to selecting the entity and clicking OK.

Back to top

11.0.130 Release Notes - MotionView Vehicle Library

TNO Magic Formula and Swift Tires Support

The TNO Magic Formula and Swift tires are now supported in the vehicle library. A tire system definition and sample TNO files are included in the installation. This requires solvers SA11.0-220.

Truck Tire System Definitions Enhancements

A series of truck tire system definitions are included to model axles with four tires. Up to 18 tires can be modeled using these system definitions.

Single Tire System Definitions

Single tire system definitions are included for vehicles. The wizard based tire system can be deleted and replaced with single tire definitions or axle pair tires. This permits easier development of vehicles with three wheels, two wheels (such as motorcycles), and other alternative suspension systems.

Vehicle Models in Online Help

The vehicle models are now documented in the online help. Documentation includes a description of the points, bodies and joints of the suspension, along with the options available with each suspension.

Resolved Issues

- The default suspension spring preloads were revised for the rear twistbeam and the rear solid axle 4 link suspensions. Vehicles built with these suspensions should now run out of the box.
- The braking event now works (requires solvers SA 11.0.230). The event would run in the past but would not apply any braking torque.
- The speed controller is revised in straightline acceleration events to use velocity control instead of acceleration control, resulting in a more robust solution.
- The “DIFF” method of tire modeling is removed from the library.
- The static load event now works for the Powertrain model when solving in MotionSolve.
- The multi leaf spring template has been fixed to allow users to customize the leaf spring configuration. The template is located in \hw\mdl\mdllib\Libs\Models\Rear_susp\Leaf_spring and allows a parametric leaf to be built.
- The report template for the “steer release in a turn” event now shows all plot results defined in the template.

- For a front half suspension the Vehicle Body can now be resolved to none and the system will still solve.
- DataSet capitalization has been revised in certain full vehicle events. Correcting the spelling will now allow datasets to be resolved by hand.
- The rear stabilizer bar connection description has been revised to match the front stabilizer bar description.
- The .MODEL. prefix has been removed from systems when building models via the assembly wizard. This partially fixes an issue that occurs when a user saves systems and then reuses them.
- Certain events have been revised to use the DefineRecord statement instead of DefineTable. DefineTable has been deprecated and should not be used.

Back to top

11.0.120 Release Notes

Modeling via Distributed Files

Through this version release, MotionView now supports modeling via distributed files. A new container entity called Assembly is introduced which has its own definition and data files. The new method would enable collaboration and sharing of aggregate sub-assembly models.

Similar concept of distributed files is also introduced for the Analysis entity. Analysis now comes in 2 flavors. The existing style is called "Inlined Analysis" and the new distributed file type is called "External Analysis."

Systems/Analysis Toolbar

The Systems/Analysis toolbar browser context has been rearranged to accommodate the new container entity Assembly. System /Assembly is a common toolbar now and Analysis has a separate toolbar.

- Adding System is integrated into the Add System/Assembly wizard style dialog
- Adding Analysis is also a wizard style dialog that includes both types of analyses.

SpringDamper Panel Enhancements

SpringDamper panel has been enhanced to have Free Length field filled in automatically as a distance between 2 end points. The panel fields also warn or error for invalid values.

Performance Improvements

CAD import utility performance has been moderately improved to handle large models.

Resolved Issues

- Finding the nodes in the Nodes panel of a deformable body could become slow when searching for a very large number of nodes. The speed has been improved.
- Deformable body graphics are now highlighted when the body is selected in browser.
- Graphics attributes will now change for a deformable body from the Graphics panel.
- Performance issue on Joint panel – changing joints performance has been improved.

- Attributes for 3D parametric curves are now correctly saved to MDL format.
- CAD import now brings in correct inertia values when importing without selecting the units in the Import dialog box.
- CG Inertia summary no longer reports erroneous inertia values when Inertia coordinate frame is different than CG.

Back to top

11.0.101 Release Notes

New Panels

New panels for all modeling entities includes hyperlinks for referenced entities, and easy navigation buttons.

Model Browser (Project Browser)

All new Model Browser (Project Browser) contains improved context menus and enhanced performance.

New HyperWorks Desktop Framework

An updated framework for the HyperWorks Desktop products gives a new look and feel geared towards improved ease of use and consistency among HyperWorks applications. The new framework provides docking toolbars (Animation, View Controls, Standard, Display, and Image Capture), and in-place command window.

Session Browser

The Session Browser is packed with powerful functionality for managing the pages and windows in a session. Page/window management of creation, reordering, deletion can be accomplished via the context menus of the Session Browser.

Saving models in sessions as Inline or Reference is enabled using the Sessions Browser. The Session Browser has an additional column to specify whether or not to save the model. This helps save all files simultaneously while saving the session file.

Context Sensitive Menu for Graphics Area

Easy select, cut and delete features using right click on the entity graphics area.

Delete Feature

New delete feature is available in both Model Browser and graphic context.

Improved File Path Management

File path management is improved using tools available under Tools > Options. This resolves some specific use cases of saving files with incorrect file references.

General Constraints Support

The GUI now supports General Constraints (Constraint_Mate model statement in MotionSolve). It is now possible to define these constraints directly in MotionView environment.

Additional H3D Output Options Supported

Output file types in the Run panel now supports additional H3D Output options for rigid and flex body outputs.

Entity Definitions Improvements

All entity definitions needing orientation now come with additional keys to the MDL syntax specifying the type of orientation. This resolves issues related to lost orientations due to unresolved references.

Saving of Properties

Saving of properties (*Set statements) is now within *BeginContext() block following a system hierarchy. In addition, concept of definition and data is introduced. See MotionView online help for more details.

Improved Contact Modeling

Contacts now have a new and improved way of modeling. Contacts can now be defined between bodies (instead of graphics) with options to chose relevant graphics associated with those bodies.

Graphical Picking Enhancements

Enhancement to graphical picking to support modular approach to model building has been added. Model tree pop-up comes with a checkbox to select items under valid scope. Using this option allows selection of dependent entities within the same system/analysis as that of entity being defined.

File Management

Open, Save, Import and Export are now supported through the File Menu and through the HyperWorks Desktop icons. These functionalities have been streamlined for better usability.

Operations related to writing out properties and Wizard Selections, which were available under Save panel earlier, are now part of Export Model operation.

Load and Save Toolbar Panels

Load and Save toolbar panels have been removed. These operations are integrated with the Open & Save Model icons (also available through the File Menu) maintaining common look and feel across other HW Desktop applications. Similarly, Import and Export of Solver Deck move to their respective HW Desktop icons.

MDL Library Updates

The MDL Library includes all 10.0 SA upgrades. The file path and extension for usersubs have been removed. This enables easy portability of run files across versions and platforms. There is no need to rename the path in the XML files when solving an XML generated in Windows and solved in Linux or vice-versa.

Resolved Issues

- Issues related to losing of *Set statements on attachments within a system/analysis upon load is fixed.

Back to top

Highlights of the HyperCrash 12.0 Release

- New Features
- Enhancements
- RADIOSS Block Format Profile
- LS-DYNA Profile
- PAM2G Profile
- Results Mapper
- Dyna2Radioss Converter

Contents

12.0 Release Notes - New Features

Part Replacement

Solid Spotweld

Quality\Contour Check

Software Failure Detection

Connection

Accurate Mass Calculation

Mass Balancing

Target Mass

Support of Stress-Strain *INITIAL

GLUE Connection

Model Checking

Edit Entities from Browser

Solver Field Names

ID Control from Generic Panel

Mandatory Field Check

Change Property and Material Connections

Xref on Picked Node and Elements

Performance

New Icons in Browser for Part

Specific Renumber

Review of Sets

Direct Selection of Sets

Element Panel Creation

Browser

12.0 Release Notes - Enhancements

Result Mapper

Penetration Check

Spotweld

Export

Replace Part

INCLUDE File

Panel Improvement

Merge Module

RADIOSS Selector

Delete Operation

12.0 Release Notes - RADIOSS Block Format User Profile

RADIOSS 11.0 Support

/UNITS

/SUBDOMAIN

Control Cards and Engine Cards

/XREF and /REFSTA

New Encryption Support

New Keywords

Modified Starter Keywords

Supported RADIOSS Keywords

12.0 LS-DYNA User Profile

*Parameter

Rigid Part Filter

*MAT_

*CONTACT_

*CONTROL

*DATABASE_HISTORY

*LOAD

*BOUNDARY

*ELEMENT

Result Initializer

*MAT_

Model Checker

*CONSTRAINED_

*ALE

*BOUNDARY

*ELEMENT

12.0 PAM2G User Profile

Keywords Improved

12.0 Results Mapper

Abaqus Reader/writer

Radioss Reader/writer

LS-Dyna Reader

New Mapping Search Algorithm

Improved Symmetry Option

12.0 Dyna2Radioss Converter

Report

Improved Conversion

New Keywords Converted

Bug Fixes

12.0 Release Notes - New Features

Part Replacement

This feature is new for LS-DYNA and enhanced for RADIOSS. The user now has a more accurate replacement covering more cards and a detailed report of what was replaced or not.

Solid Spotweld

Solid spotweld connection is now available for RADIOSS.

Solid spotwelding can also be modeled using a cluster of 4, 6, 8, 12 and 16 combined solids, in addition to a single HEXA for RADIOSS [starting with 10.0-SA-205] and LS-DYNA.

Quality/Contour Check

Quality\Contour check uses colors to differentiate between the parts based on different attributes like Young module, density, yield stress and more.

Software Failure Detection

Great effort has been taken to protect the software. When you restart HyperCrash after a bug detection or failure, the saved session can be restored at start-up.

Connection

Offer parameterization power for element size, failure criteria and contact used for both 1D and 3D welds.

Accurate Mass Calculation

In LS-DYNA a new mass calculation is available to match the values of the solver.

Mass Balancing

Now masses can be distributed to match user defined values of Target Mass and Center of Gravity.

HyperCrash has the ability to create mass of general types or special ones used to represent a missing system in the model or masses created during the balancing process.

Target Mass

With an external XML file, the user can assign target mass to multiple parts simultaneously.

Support of Stress-Strain *INITIAL

LS-DYNA now supports the Stress-Strain *INITIAL conditions for all element types.

GLUE Connection

Initial support of GLUE (mastic) connection defined using continuous HEXA is now available.

Model Checking

Model accuracy is enhanced by real time checks of the mandatory fields essential to run an analysis.

Edit Entities from Browser

Entities can be edited directly from the browser. In previous versions, you could only edit from the appropriate entity creation pages.

Solver Field Names

In the generic pages we now show solver field names as specified in the solver manual.

ID Control from Generic Panel

It is now possible to set the max ID of the model or the first available ID at the time of entity creation.

Mandatory Field Check

When creating new entities, the mandatory fields are highlighted in red. Also, a check is performed to see if the input value is in range accepted by the solver.

Change Property and Material Connections

Properties and Materials of several connections can now be changed in one step. Connections > Spotwelds > Modify and select the connections.

Xref on Picked Node and Elements

When an entity is picked on the screen, Xref information is posted in a new window. The node and element information is still posted in the message window as in previous versions.

Performance

Improvements have been made in ModelChecker and Check Connectivity features.

New Icons in Browser for Part

New icons have been added in the Parts folder to show if a part is empty and parts that have forming results applied.

Specific Renumber

Part and Assembly are now supported under specific renumber.

Review of Sets

Sets can be reviewed from the browser.

Direct Selection of Sets

Instead of editing/selecting graphic entities, it is possible to directly select an existing Set/Group.

Performance

Improvement in file reading through a network.

Element Panel Creation

Under Mesh Editing menu there is now access to new menu entries for opening new panel enabling the user to create elements, including the non connectivity fields.

Browser

Part/Assembly mode

Show/hide of connection linked options

Back to top

12.0 Release Notes - Enhancements

Result Mapper

Enables the Results Mapper in LS-DYNA profile.

Penetration Check

Proper coverage of an LS-DYNA penetration check.

Spotweld

Spotweld can be optionally created using a single material and property, instead of individual ones for each logical connection.

Export

Improvement in the GUI of the export selection is now offered.

For LS-DYNA, HyperCrash exports at the end of the deck a block of information used to preserve target mass and COG to import / export.

Replace Part

When performing a Replace Part for RADIOSS, /SECT are taken into account.

INCLUDE File

For RADIOSS and LS-DYNA a feature is now available to control RELATIVE/ABSOLUTE PATH for Include.

For LS-DYNA, HyperCrash now allows a long directory path for INCLUDE files.

Proper support for *INCLUDE_STAMPED_PART and *INCLUDE_NASTRAN.

Panel Improvement

For LS-DYNA, improvement of the BOX panel is now offered.

For RADIOSS, enhancement in specific renumbering for part(s). Part and subset are no longer grouped together in the renumbering panel.

For all LS-DYNA panels and selected RADIOSS panels:

- Dynamic review is now available in a list using the up and down arrows on the keyboard.
- Multiple items can be selected holding the left mouse and scrolling the mouse.
- A new review capability is available using the right-click option.

Merge Module

Add option to merge an Include or include transform of an imported file.

RADIOSS Selector

Validation by right-clicking the mouse reduces mouse travel by offering the completion of the process, such as selection of items within the graphics area. This is in addition to clicking the OK button or pressing the ‘Y’ key on the keyboard.

Delete Operation

A faster and more accurate delete operation is now offered.

Back to top

12.0 Release Notes - RADIOSS Block Format User Profile

RADIOSS 11.0 Support

New RADIOSS profile has been added to support RADIOSS 11.0. The supported keywords are listed below.

/UNITS

It is now possible to have models of different units in the same session. Entities can have their own units system as allowed by RADIOSS.

/SUBDOMAIN

This card for defining Subdomains is now supported: read, export and creation/edition is allowed.

Control Cards and Engine Cards

Control and Engine cards are exposed under Model drop-down and shown in browser.

/XREF and /REFSTA

/XREF and /REFSTA cards are now supported for Reference geometry.

New Encryption Support

RADIOSS 11.0 supports new encryption keywords, which are supported in this version of HyperCrash.

New Keywords

/ATFILE

/FAIL/XFEM_FLD

/FAIL/XFEM_JOHNS

/FAIL/XFEM_TBUTC

/RFILE/n

/STATE/DT

/TFILE

/MAT/LAW72

Initial state keywords

/INIVEL/AXIS

/SUBDOMAIN

/CONVEC

/ADMESH/… cards

/GAUGE and related keywords

/PROP/PLY, /PROP/STACK

/BOX

/RADIATION

Modified Starter Keywords

/DAMP

/INTER/TYPE21, /INTER/TYPE20

/VISC/PRONY

/MAT/LAW19, /MAT/LAW27, /MAT/LAW58

/INITEMP (RADIOSS 10.0 Starter)

/MAT/LAW18

/INTER/TYPE2

Supported RADIOSS Keywords

/MAT/LAW77, /MAT/LAW70, /MAT/LAW69, /MAT/LAW73

/FAIL/TAB, /FAIL/XFEM_FLD

Initial state keywords

/INITEMP (RADIOSS 10.0 Starter)

/INIVEL/AXIS

/RBE2

/EOS/TILLOTSON, /EOS/GRUNEISEN, /EOS/POLYNOMIAL

/PROP/TYPE26

/MPC

/EBCS

Back to top

12.0 Release Notes - LS-DYNA User Profile

***Parameter**

*Parameter is supported in this version.

*Parameter expression will be supported in a future version.

Rigid Part Filter

During entity creation that requires rigid parts, for example constrained_rigid_bodies, we now filter the part list to show only the rigid parts that are available for selection.

***MAT_**

MAT_072R3, MAT_125, MAT_134, MAT_151, MAT_153, MAT_155, MAT_156, MAT_159, MAT_159_CONCRETE, MAT_164, MAT_165, MAT_166, MAT_167, MAT_168, MAT_171, MAT_172, MAT_173, MAT_174, MAT_175,

MAT_LOW_DENSITY_SYNTHETIC_FOAM_ORTHO_WITH_FAILURE (180),

MAT_ADD_PERMEABILITY and MAT_ADD_THERMAL_EXPANSION supported

***CONTACT_**

…_DAMPING option taken into account

CONTACT_TIEBREAK_SURFACE_TO_SURFACE_ONLY, some fixes.

***CONTROL**

CONTROL_IMPLICIT_CONSISTENT_MASS,

CONTROL_IMPLICIT_FORMING,

CONTROL_IMPLICIT_STATIC_CONDENSATION,

CONTROL_MPP_DECOMPOSITION_BAGREF,

CONTROL_MPP_DECOMPOSITION_DISTRIBUTE_ALE_ELEMENTS,

CONTROL_MPP_DECOMPOSITION_DISTRIBUTE_SPH_ELEMENTS,

CONTROL_MPP_DECOMPOSITION_ELCOST,

CONTROL_MPP_DECOMPOSITION_OUTDECOMP,

CONTROL_MPP_DECOMPOSITION_PARTS_DISTRIBUTE,

CONTROL_MPP_DECOMPOSITION_SCALE_CONTACT_COST,

CONTROL_MPP_DECOMPOSITION_SCALE_FACTOR_SPH,

CONTROL_MPP_IO_BINOUTONLY,

CONTROL_MPP_IO_LSTC_REDUCE

***DATABASE_HISTORY**

_SPH, _SPH_SET, _TSHELL supported in I/O

***LOAD**

LOAD_BODY_GENERALIZED_SET_NODE, LOAD_BODY_GENERALIZED_SET_PART, LOAD_THERMAL_LOAD_CURVE, LOAD_THERMAL_VARIABLE_SHELL, LOAD_THERMAL_VARIABLE_SHELL_SET

***BOUNDARY**

BOUNDARY_SPC_NODE_BIRTH_DEATH, BOUNDARY_SPC_SET_BIRTH_DEATH (_ID as well for both)

***ELEMENT**

_DISCRETE_LCO, _SPH now supported in I/O

Result Initializer

This tool enables to compute the initial stresses/plastic strains and thicknesses using a one-step approach. So INITIAL_STRESS_SHELL/ELEMENT_SHELL_THICKNESS cards are created. Needs to be installed with a HM11.101 version minimum.

A part from this tool it is now possible to display initial results from forming loaded with the model.

***MAT_**

MAT_113, MAT_131,MAT_132, MAT_173

Model Checker

Rigid Body defined on Rigid body node (MAT_RIGID, Extra_node, NRB, Generalized_weld, etc.)

In Extra_node, PID or NID/NSID is Zero

In Extra_node, PART is not Rigid

Joint used non rigid body nodes

Joint is not connected to any parts

-*MAT and/or *SECTION not referred by PART

Double(same) ID in table and curve

Number of curve is not same as one defined by table

Same part are defined in slave and master in face to face contact

Too many lines in *SET_SHELL_GENERAL, *SET_SEGMENT_GENERAL -> Warning in case of more than 100

Check double Id for every type of entity which requires an Id

Slave set of *CONTACT as null should create a error.

***CONSTRAINED_**

*CONSTRAINED_TIED_NODES_FAILURE, *CONSTRAINED_LAGRANGE_IN_SOLID

***ALE**

*ALE_FSI_PROJECTION

***BOUNDARY**

BOUNDARY_NON_REFLECTING

***ELEMENT**

New panel for creating elements (*ELEMENT_...) including the non-connectivity fields

Back to top

12.0 Release Notes - PAM2G User Profile

Keywords Improved

Improvement of CNTAC support

*MAT_196, *LOAD_RIGID_BODY, *MAT_067

Back to top

12.0 Release Notes - Results Mapper

Abaqus Reader/writer

Supported type for NODES :

CONCENTRATION

FLUID PRESSURE

PRESSURE STRESS

RELATIVE DENSITY

SATURATION

TEMPERATURE

VELOCITY

Supported type for elements:

PLASTIC STRAIN

STRESS

Radioss Reader/writer

INITEMP (temperature on nodes)

INIBRI/STRA_F card (strain tensor on solids)

INISHE/ORTHLOC and INISH3/ORTHLO cards (fiber or orthotropic directions)

LS-Dyna Reader

*INITIAL_STRAIN_SHELL

*INITIAL_STRAIN_SOLID

*INITIAL_STRESS_SOLID

*INITIAL_STRESS_TSHEL

New Mapping Search Algorithm

Memory used and speed improvement

Neighbor search area size is now computed from the forming model mesh element sizes and not any more from the target mesh element sizes => speed and quality improvement

Improved Symmetry Option

Forming results are now duplicated and then mapped which is more accurate. On previous versions, mapping results were mapped first and then mapped results were mapped again as many time as there were symmetries.

Back to top

12.0 Release Notes - Dyna2Radioss Converter

Report

New html report of conversion

Improved Conversion

*CONTACT_INTERIOR, *DATABASE_CROSS_SECTION_PLANE, *MAT_024, *MAT_003

*INITIAL_VELOCITY (Boxes referenced properly taken into account), *RIGID_WALL (Boxes referenced properly taken into account), *MAT_068, *ELEMENT_BEAM associated with *MAT_066, *MAT_029, *MAT_020, *AIRBAG_..., *DEFINE_BOX, *DATABASE_RWFORC, *SET_PART_LIST,

New Keywords Converted

*MAT_018, *MAT_032, *MAT_026, *MAT_126, *MAT_DAMPER_NONLINEAR_VISCOUS, *MAT_NONLINEAR_PLASTIC_DISCRETE_BEAM, *MAT_DAMPER_VISCOUS, *ELEMENT_SEATBELT_SLIPRING, *AIRBAG_REFERENCE_GEOMETRY

*MAT_196, *LOAD_RIGID_BODY, *MAT_067

Bug Fixes

HyperCrash no longer crashes while replacing a part with RBODY created with BOX.

HyperCrash now correctly invokes the help from the local installation instead of automatically opening its web version.

INCLUDE isolate in browser now working in both LS-DYNA and RADIOSS.

Improvement in LS-DYNA find connectivity and other special conditions are covered.

Back to top

HyperWorks Solvers 12.0 Release Notes

HyperWorks Solvers is a collection of finite element and multi-body dynamics solvers for structural, fluid-dynamics and systems simulation. These solvers can be employed for simulation driven design as well as design optimization.

RADIOSS	Finite element solver for linear and nonlinear problems
AcuSolve	General purpose finite element computational fluid dynamics (CFD) solver
MotionSolve	Multi-body dynamics solver
OptiStruct	Design and optimization software using finite elements and multi-body dynamics
HyperForge Solver	New solver targeted for extrusion related open die forging applications
HyperXtrude-RTM Solver	Finite element based solver for analysis and design of resin infusion processes

Folder Structure Reorganization

The folder structure in the installation directory has been modified for HyperWorks 12.0 as compared to HyperWorks 11.0. The average user will not be affected by this reorganization if the GUI is used to launch the solvers. Advanced users, who invoke the solvers from the command line, build user subroutines or use custom scripts which call the solver executables directly should look at the changes listed in Table 1.

Table 1: Folder Structure Reorganization

	11.0	12.0

Invoke Point
For users who invoke from the command line
	Unix/Linux/Mac:–
	`altair/scripts/<solver>`
	Windows:
	`32 bit: altair/hwsolvers/bin/win32/<solver>.bat`
	`64 bit: altair/hwsolvers/bin/win64/<solver>.bat`
		Unix/Linux/Mac:
		`altair/scripts/<solver>`
		Windows:
		`Altair/hwsolvers/scripts/<solver.bat>`

Usersubs (MotionSolve) For users who build user subrouties for MotionSolve	`Altair/hwsolvers/usersub`	`Altair/hwsolvers/motionsolve/usersub`
Executables For users who use custom scripts that call the solver executables directly, like cluster	`Altair/hwsolvers/bin/<platform>`	`Altair/hwsolvers/<solver>/bin/<platform>`

Licensing Libraries/Paths
For users who use custom scripts that call the solver executables directly, like cluster
	`RADFLEX_PATH=altair/hwsolvers/bin/<platform>`	`RADFLEX_PATH=altair/hwsolvers/common/bin/<platform>`
		Windows:
		`PATH = altair/hwsolvers/common/bin/<platform>`
		Non-Windows:
		`LD_LIBRARY_PATH=altair/hwsolvers/common/bin/<platform>`

Highlights of the MotionSolve 12.0 Release

The 12.0 release of MotionSolve contains significant enhancements in core modeling, solutions methods and extensions tailored for the automotive industry. These changes provide new modeling and analysis functionality, enhance software robustness and offer delightful simulation speed benefits. Some of the key improvements are noted below.

- Accuracy and Performance Improvements For Rotating Bodies
- Flex-Body Improvements
- Improved Contact Modeling And Solution
- Joint Friction
- Enhanced Linear Analysis
- Comprehensive Tire and Road Support
- Powertrain Model For Use in Vehicle Dynamics

Contents

Core Functionalities

- ABF File Enhancements To Support Downstream Fatigue Calculations
- ADAMS DLL Compatibility
- AeroDyn Libraries are Now Supported
- Deformable Curves and Surfaces
- Distributed Forces on Flexible Bodies
- Enhanced Linear Analysis
- Flex-Body Improvements
- Friction and Normal Force Definition for Force PTdSF through User-Subroutines
- Improved Contact Modeling And Solution
- Improved General State Equation Modeling Element
- Initial Conditions Improvements
- Joint Friction
- MotionSolve--Simulink Co-Simulation
- New Optimization Functions
- Performance Improvements For Rotating Bodies
- Python 2.7.3 Supported
- Save/Reload Improvements
- Version Of Binary Files

Automotive Extensions to the Solver

- CD-Tire 3.0.3 Support
- Dynamic Loading of Cosin Tire Interface Library
- Enhanced Methodology For Defining Vehicle Velocities
- RMOD-K Tire 709 Tire Model
- Powertrain Model For Use in Vehicle Dynamics
- Static Design Factors Improvement
- Tire Free Rolling (Angular) Speed
- Tire Utilities
- TNO MF And TNO SWIFT Tire Model Support

Software Behavior Changes

- User Subroutine Search Behavior
- Warning for Special Characters

ADAMS Compatibility

- ADM Compatibility
- ACF Compatibility
- User Subroutine Compatibility

12.0 Resolved Issues

- ADM File Import
- Contact Force Graphics
- Deformable Surface
- Flex-bodies
- H3D Animation Speed In 11.0
- HTire Tire Model Licensing
- Initial Velocity Calculations
- Motion Expression
- Offset Issues when Flexible Marker Location & Node Position are Not Coincident
- Parametric Curve
- Post_Request Initialization
- Python Splines
- Run Termination
- Unexpected Euler Angle Rotations for Markers During Static Equilibrium
- SAVE/RELOAD: Inconsistent Velocities After a SAVE/RELOAD Operation

- SAVE/RELOAD: Output Not Generated for RELOAD Operations
- SAVE/RELOAD: Unexpected Orientations After a SAVE/RELOAD Operation
- Support for Multiple PINPUT and POUTPUT in the XML File
- Units Support For Damping In Bushing Property Files

Introduction

MotionSolve is a state-of-the-art multi-body solver available in HyperWorks. It has a complete set of modeling elements and powerful numerical methods to support a full set of analysis methods. The accuracy, speed and robustness of MotionSolve have been validated through extensive testing with customer models and test data. MotionSolve also offers unmatched compatibility with ADAMS/Solver input. Click here for more information about MotionSolve.

This document describes changes that have occurred to MotionSolve since November 2011. All of these improvements are available in HyperWorks 12.0.

Accuracy and Performance Improvements for Rotating Bodies

Speeds in excess of 3X and improved accuracy have been seen for many full vehicle handling simulations. This improvement has been accomplished primarily by changing the equation formulation methods in MotionSolve to account for the high rotation speeds of components such as tires in such models.

Flex-Body Improvements

The efficiency of Component Mode Synthesis (CMS) in RADIOSS for generating modal flex-bodies has been improved. Speeds improvements in excess of 8X have been observed when performing CMS for large systems such as the Body-In-White (BIW) component for automobiles. A capability for applying distributed loads on flexible bodies is also introduced.

Improved Contact Modeling and Solution

The calculation of contact points, penetration depth and contact normal has been significantly improved for discrete geometries such as tri-meshes. Contacts forces are now smoother and the effects of geometry discretization are reduced. Specifically, these improvements are seen for rigid bodies undergoing persistent, sliding contact. Smoother forces have also resulted in more robust solutions and fewer failures.

Joint Friction

A new modeling entity Force_JointFriction is available for modeling stiction and dynamic friction in joints. The well-known LuGre model is used to define friction behavior. Frictional effects due to preload, joint reaction forces and moments and damping are considered in the formulation. Detailed joint geometry is used to identify the mating surfaces and apply the friction effects.

Enhanced Linear Analysis

Linear analysis in MotionSolve has been completely redone to support all modeling elements and make it more stable. Through a linear analysis at an operating point, users may examine system vibration modes and frequencies or extract system transfer functions as ABCD matrices.

Comprehensive Tire and Road Support

All major tire models are now supported. Standard implementations of Pacejka models for vehicle handling and higher fidelity models with flexible belt models suitable for NVH, Ride and Durability are now available. In addition to F-Tire, MotionSolve now supports CD-Tire 3.0.3, RMOD-K 709, TNO MF and TNO/Swift tire models.

Powertrain Model for Use in Vehicle Dynamics

A simplified and efficient yet accurate equation-based powertrain model, suitable for vehicle models undergoing handling and durability events, is now available. The underlying equations accurately capture the behavior of the internal combustion engine and the clutch in the powertrain.

Back to top

Core Functionalities

ABF File Enhancements to Support Downstream Fatigue Calculations

ABF is an Altair proprietary binary file format that may be used to store MotionSolve simulation results. A new attribute in the XML input deck for the ResOutput statement now ensures that for each output time, only one set of simulation results is stored – the latest set of results. Previously, if a model was modified at time=Tn and the simulation continued, two sets of results were output. The first was for time=Tn prior to the model change and the second for time=Tn after the model change. While this earlier approach was valid and meaningful, it causes difficulty to fatigue solvers that requires uniqueness in their time series signals. The XML snippet for controlling unique time outputs is shown below.

Enable unique time output in ABF file. Default is “FALSE”.

```
<ResOutput
    abf_unique_time = "TRUE"
/>
```

This option will make sure there is only one output per time in the ABF file. If multiple outputs are generated at the same time during the solver run, the latest data will be used.

Adams DLL Compatibility

ADAMS/Solver DLLs, using the public subroutine interface of ADAMS/Solver will automatically work with MotionSolve. Existing user subroutines compiled for ADAMS and available as DLLs or shared libraries can now simply be reused with MotionSolve. MotionSolve is able to work with these DLLs, provided the publicly documented user-subroutine interface is strictly adhered to. There is no need to recompile source code or re-link the object files.

AeroDyn Libraries are Now Supported

MotionSolve now supports the AeroDyn libraries for aero-elastic analysis of wind turbines. The National Renewable Energy Laboratories (NREL) originally developed AeroDyn, an aerodynamics software

library for use by designers of horizontal-axis wind turbines. The aerodynamics model in AeroDyn is a detailed analysis that includes blade-element/momentum theory (with modifications to improve accuracy in yawed flow), dynamic stall, and an optional dynamic inflow theory. AeroDyn has been tested with models published by NREL, and NREL has certified its use with MotionSolve.

AeroDyn libraries are now included in the MotionSolve installation at `<altair_root>\12.0\hwsolvers\motionsolve\bin\<platform>`.

Deformable Curves and Surfaces

Deformable curves, used by the Point-to-Deformable-Curve constraint entity and deformable surfaces, used by the Point-to-Deformable-Surface constraint, can now be open or closed. This allows for a continuous curve/surface description where the constraint can apply without interruption.

Distributed Forces on Flexible Bodies

A new capability for applying distributed forces on a Flex_Body through a new FORCE_MODAL modeling element is now available. The distributed force on the Flex_Body is defined in the modal domain and not in physical coordinates.

Through the use of this strategy, the potentially expensive physical to modal conversion of the force is done only once. However, this also makes the element somewhat challenging to use. Users have to define the distributed load in a finite element modeler like HyperMesh. The model is then transferred to a finite element solver, such as RADIOSS, where the conversion from physical to modal coordinates is done as a part of the component mode synthesis and generation of the Flex_Body.

Distributed forces are particularly useful for modeling phenomena such as aerodynamic loads, temperature loads, pressure loads, state dependent damping and even limited forms of contact involving flexible bodies.

There are several alternative ways in which the distributed load can be defined in MotionSolve. Some of these use the expression language to define simpler, time-dependent spatial loads whereas others use the MFOSUB user-subroutine to define more complex scenarios. Please refer to the reference documentation on FORCE_MODAL for a more thorough explanation.

Enhanced Linear Analysis

Linear analysis in MotionSolve has been significantly enhanced. It now supports all modeling elements. Through a linear analysis, users may examine system level vibration modes or extract Multiple Input-Multiple Output transfer functions as ABCD matrices for any operating point in the analysis.

Users can control the size of their output files by optionally disabling damping and modal output to the mrf file, if these data blocks are not of interest.

Linear analysis animation results are now stored in the MotionSolve H3D animation result file, which can be directly viewed in HyperView. All modal viewing capabilities of HyperView are available. The H3D file supports the case where multiple linear analyses are performed during a single time domain analysis.

Flex-Body Improvements

The Flex-Body capability in MotionSolve has been improved in several ways.

The efficiency of component mode synthesis methods in RADIOSS for generating modal flex-bodies has been significantly improved. **Speed improvements in excess of 8x have been observed when performing component model synthesis for large systems such as the Body-In-White (BIW) component for automobiles.** This improvement is seen for both the Craig-Bampton and the Craig-Change methods. The Craig-Bampton method is now much faster for problems with a large number of interface nodes (ASET DOF). The Craig-Chang method is dramatically faster when the AMSES or AMLS eigenvalue methods in RADIOSS are used.

Flex-body component and assembly information is now transferred all the way from HyperMesh via RADIOSS and MotionSolve to HyperView. Flex-body component and assembly information is now written to the MotionSolve output H3D file. This enables visibility of flex-bodies to be controlled by component or assembly in HyperView.

Nodal velocities and accelerations are now written to the MotionSolve H3D output file. This change allows velocities and accelerations to be visualized in HyperView.

The search mechanism for finding Flex-H3D files has been improved. MotionSolve now allows you to specify an environment variable to define a set of paths to search for flex-body files. This provides you with the capability to have a centralized repository of flex-bodies that can be easily used by models located on various areas of your computer. To use this feature, specify the environment variable `MS_H3DFILE_DIR` and point it to a list of the desired search folders separated by the ";" delimiter. MotionSolve will now search for the flex h3d (with relative path, if specified), at the following locations (in order) till it is found:

- Input file folder
- Each folder location in the list specified by the `MS_H3DFILE_DIR` environment variable

H3D files have been enhanced to contain a new shell thickness attribute. This additional piece of information is necessary to be able to calculate fatigue or damage of shell structures such as automotive bodies or automobile doors.

Friction and Normal Force Definition for Force PTdSF through User-Subroutines

The Force_PTdSF element defines a contact force between a rigid body and a deformable surface. This modeling element has been enhanced to support user-defined models for contact normal force and frictional force. A user subroutine, PTDSFSUB, is required to provide the user-defined contact models. PTDSFSUB may be written in Fortran, C or C++ languages, or alternatively in Python or Matlab scripts.

The output signals from a Force_PTdSF that are written to the MotionSolve Results File (MRF) and Altair Binary File (ABF) have been extended to include the following time based quantities:

- The X-Y-Z locations of the contact point
- The X-Y-Z components of the surface normal at the contact point
- The penetration depth
- The contact normal force (which acts along the surface normal)
- The X-Y-Z components of the friction force vector
- The U and V parameter values for the surface at the contact point
- The tracking error, a measure of how accurately the contact point has been determined

The components of all vector quantities are defined in the coordinate system in which the surface is defined. These quantities can now be plotted in HyperGraph.

A family of Force_PTdSF elements may be used to model contact between two deformable surfaces. However, there must be only one contact point between the two surfaces. One common application is to analyze moan and squeal phenomena in disk brakes.

Improved Contact Modeling and Solution

The 3-D contact capability in MotionSolve has been improved in several ways.

The calculation of contact points, penetration depth and contact normal has been significantly improved for discrete geometries such as tri-meshes. Specifically, these improvements are seen for rigid bodies that are undergoing persistent, sliding contact. As a result of these improvements, contacts forces are smoother and the effects of geometry discretization have been reduced. Figure 1 below shows the improvement in the contact results. Notice that the new results are smoother than the "old" results.

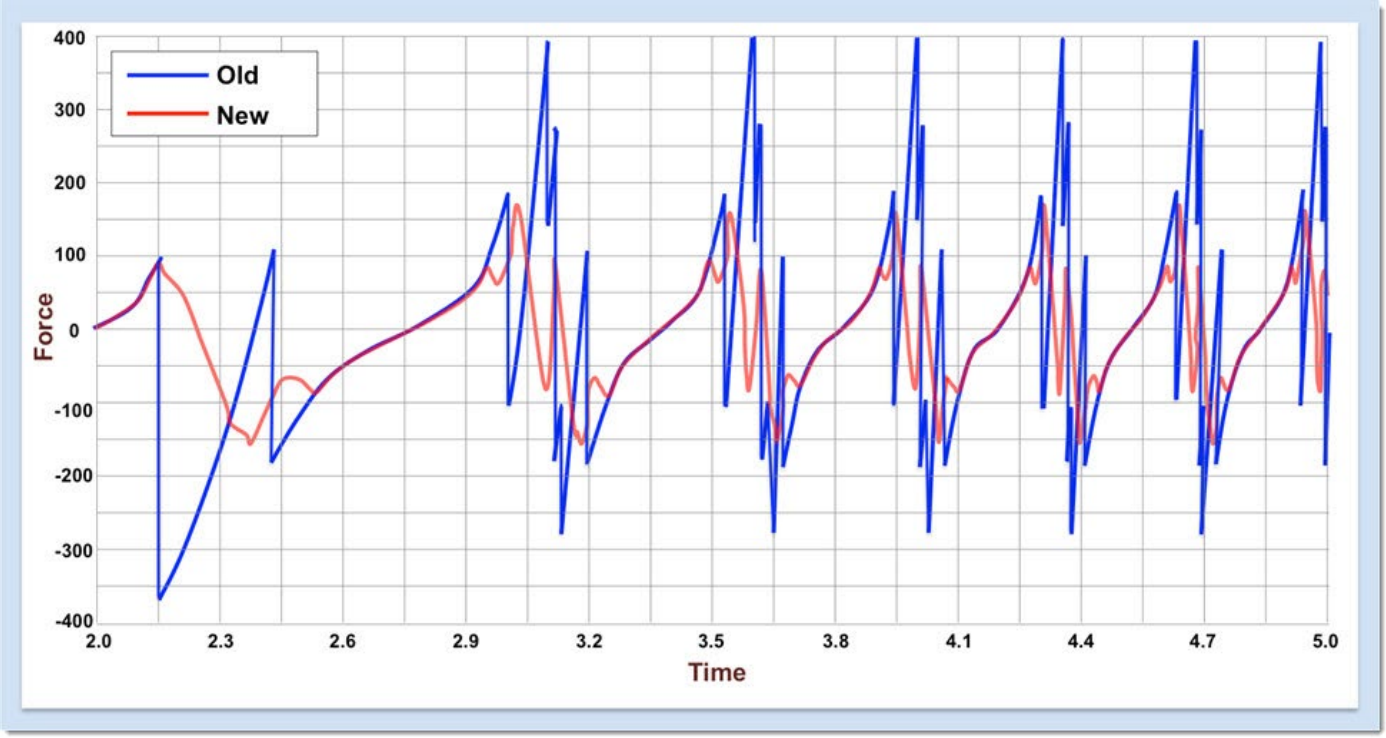


Figure 1: Contact forces for 12.0 are smoother

The visualization of contact forces in HyperView has been significantly improved. For any given contact definition, normal and tangential forces for each of the contact locations are now optionally saved in the MotionSolve H3D output file. These forces can subsequently be viewed for each time frame in the simulation in HyperView. A new *contact_gra_output* attribute was added to the *ResOutput* command to enable this feature.

Improved General State Equation Modeling Element

The General State Equation (GSE) element has been enhanced to support DAE descriptions. *SET_GSE_ALGEBRAIC_EQN* is provided as new callable utility subroutines for defining DAE. This extends the range of phenomena that can be modeled with a *Control_StateEqn* (GSE) entity in MotionSolve.

The entity has also been enhanced to support sparse partial derivative matrices. Thus the matrices returned by the user subroutines *GSEXX*, *GSEXU*, *GSEYX* and *GSEYU* can be sparse. A new utility, *SET_GSE_NONZERO_ENTRY* is provided for defining the non-zero entries of the GSE derivative matrices in a sparse form. This method will yield significant performance improvements when the derivative matrices are sparse, i.e. most of the entries are zero.

Initial Conditions Improvements

For transient analysis of models containing initial constraint violations, the initial assembly is significantly more robust.

Joint Friction

Joint friction is now supported as a model statement, command and function. A new modeling entity, *Force_JointFriction* is now available in 12.0. Joint friction allows you to model stiction and dynamic friction in the motion of one or more directions in a joint. Supported joints include revolute, spherical, translational, cylindrical, and universal. The LuGre model is used to define the friction behavior. Frictional effects due to preload, joint reaction forces and moments are considered in the formulation. Joint geometry is used to define the mating surfaces. Normal forces on the mating surfaces are used to compute the resultant friction forces.

Friction is modeled as the average deflection force of elastic springs between two mating surfaces. When a tangential motion is applied, the bristles will deflect like springs. If the deflection is sufficiently large, the bristles start to slip. The slip velocity between the mating surfaces, vslip, determines the average bristle deflection for a steady state motion. Figure 2 below depicts the bristle model.

The LuGre model is capable of representing several different effects:

- The effect of the mating surfaces being pushed apart by lubricant
- The Stribeck effect: at very low speed. When partial fluid lubrication exists, contact between the surfaces decreases and thus friction decreases exponentially from stiction.
- Rate dependent friction phenomena such as varying break-away force, frictional lag and frequency of the input
- Static friction between two surfaces

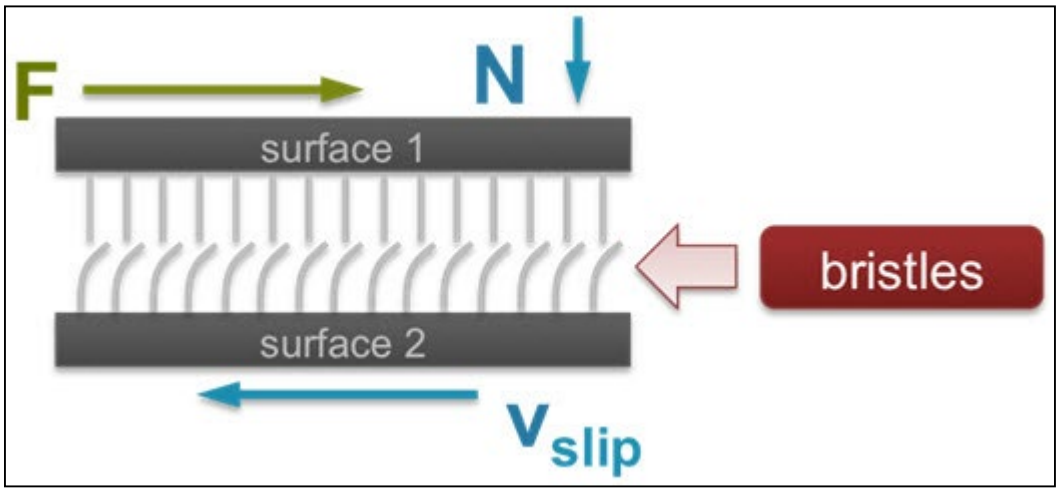


Figure 2: The bristle model for friction

A new *Friction* function is now available in the expression language. A variety of friction results are accessible through this function. The *Friction* function may be used in expressions in *Post_Request* to generate desired friction output in results files.

The *Force_JointFriction* command can be used to modify the parameters of the *Force_JointFriction* model element at run time.

MotionSolve - Simulink Co-Simulation

Co-simulation between MotionSolve and Simulink no longer need to be on the same machine or even on the same process. Co-simulation through TCP/IP sockets is now available. This means that the computer hosting MotionSolve can be geographically separated from the computer hosting Simulink. This is easily accomplished by associating IP addresses with the input signals to MotionSolve and Simulink.

New Optimization Functions

MotionSolve now supports an optimization algorithm, FMIN_SLSQP as a utility subroutine that may be called from any MotionSolve user subroutine. FMIN_SLSQP is the interface function for the SLSQP optimization function originally implemented by Dieter Kraft. SLSQP is a Sequential, Least SQuares Programming algorithm.

Optimization can be done in MotionSolve through the use of a CONSUB user subroutine together with FMIN_SLSQP. This can be used for model identification or mechanism optimization. The CONSUB may be written in Fortran/C/C++ or Python/Matlab scripts.

Performance Improvements for Rotating Bodies

Both the accuracy and performance of MotionSolve have significantly improved for models containing components that have a relatively high rate of rotational speed. **Speeds in excess of 3x and improved accuracy have been seen for many full vehicle handling simulations.** This improvement has been accomplished primarily by changing the equation formulation methods in MotionSolve to account for the high rotation speeds of components in such models.

Python 2.7.3 Supported

MotionSolve user-subroutines can be written in Python. This facility allows users to describe complex phenomena in a powerful scripting language. In 12.0, Python 2.7.3 is supported. This upgrade allows two major Python packages, NumPy (http://numpy.scipy.org/) and SciPy (http://scipy.org) to be used in the user subroutines.

NumPy is the fundamental package for scientific computing with Python. It contains among other things:

- A powerful N-dimensional array object
- Sophisticated (broadcasting) functions
- Tools for integrating C/C++ and Fortran code
- Useful linear algebra, Fourier transform, and random number capabilities

Besides its obvious scientific uses, NumPy can also be used as an efficient multi-dimensional container of generic data. Arbitrary data-types can be defined. This allows NumPy to seamlessly and speedily integrate with a wide variety of databases. NumPy is licensed under the BSD license, enabling reuse with few restrictions.

SciPy is a collection of mathematical algorithms and convenience functions built on the NumPy extension for Python. It adds significant power to the interactive Python session by exposing the user to high-level commands and classes for the manipulation and visualization of data. With SciPy, Python transforms into a data processing and system-prototyping environment rivaling systems such as MATLAB.

The additional power of using SciPy within Python, however, is that a powerful programming language is also available for use in developing sophisticated programs and specialized applications. Scientific applications written in SciPy benefit from the development of additional modules in numerous domains created by developers across the world. Everything from parallel programming to web and database subroutines and classes have been made available to the user. All of this power is available in addition to the mathematical libraries in SciPy.

This power is now available for use within MotionSolve user-subroutines.

Save/Reload Improvements

SAVE_MODEL and RELOAD_MODEL are provided as new callable utility subroutines. These can be used in conjunction with the utility optimization subroutine FMIN_SLSQP for faster system optimization. It eliminates the need to perform multiple simulations.

Version of Binary Files

To assist with providing more detail of specific binaries used by MotionSolve, detailed version information about any binary used is now printed into the MotionSolve log file.

Back to top

Automotive Extensions to the Solver

CD-Tire 3.0.3 Support

MotionSolve now supports CDTire version 3.0.3 from Fraunhofer LBF. CDTire is suitable for comfort and durability applications, but it can also be used for handling maneuvers. Depending on the application area, there are three basic CDTire sub-models that the user can select:

- **CDTire 20** – a model with rigid belt suited for handling applications on smooth surfaces
- **CDTire30** – a flexible belt tire with a brush type contact and ability to model stick-slip behavior. Suitable for in-plane only obstacle enveloping.
- **CDTire40** – a 3D flexible shell carcass with a brush type contact and ability to model stick-slip behavior. Suitable for full obstacle enveloping.

CDTire also offers two transitional sub-models. The sub-model transition is defined in the tire control file.

- **CDTire2030** – transitional model between 20 and 30
- **CDTire2040** – transitional model between 20 and 40

CDTire needs tire and road property files. The generation of tire property files requires a set of force/moment measurements, as well as topology data, cleat measurements.

CDTire supports various road property files, including digitized road data surfaces and parametrically described road surfaces, as well as user defined road models. The supported road files are categorized, as follows:

- **1000** – parametric road surface description
- **1002** – rolling drum road surface
- **1100** – user defined road
- **2000** – parametric and digitized road data in binary format

For more information about the various road models, please consult the CDTire documentation. A license to enable CD-Tire can be obtained from Altair.

Dynamic Loading of Cosin Tire Interface Library

CTI Functions (for HTire and FTire tire models) are now accessed via function pointer allowing dynamic loading of the CTI library. This means that it is now possible to use any CTI library developed by COSIN directly in MotionSolve.

Enhanced Methodology for Defining Vehicle Velocities

Two new Control subroutines are now available to easily set initial velocities for full vehicle models.

The first UserProgramControl command can be used to set wheel angular velocities.

```
<UserProgramControl
   usrsub_param_string = "user(0,30000)"
   usrsub_dll_name     = "mbdtire"
   usrsub_fnc_name     = "setwic"
/>
```

The parameter list contains two arguments:

- P1 is reserved for future use
- P2 is the vehicle longitudinal velocity expressed in model units along the x direction of the tire I marker reference frame

The UserProgramControl command will detect all the tire forces defined in the model and will find the appropriate part (usually the wheel) on which the tire force acts. The wheel initial angular velocity about the spin axis will then be computed. This should be done between the static equilibrium analysis and the transient analysis. During the calculation of the wheel omega, the actual loaded radius is taken into account in order to minimize the initial longitudinal slip.

The second UserProgramControl can be used to set body longitudinal velocities. The syntax for the command is:

```
<UserProgramControl
   usrsub_param_string = "user(0, 0, -30000, 0, 0)"
   usrsub_dll_name     = "mbdtire"
   usrsub_fnc_name     = "setic"
/>
```

The parameter list contains 5 mandatory arguments:

- P1 is reserved for future use
- P2 is reserved for future use
- P3 is the model desired longitudinal velocity in global X reference frame
- P4 is the model desired longitudinal velocity in global Y reference frame
- P5 is the model desired longitudinal velocity in global Z reference frame

An optional 6th argument can be specified. It corresponds to an array ID containing a list of rigid bodies. These body velocities will not be set during the UserProgramControl command. Typically bodies that are skipped are roads, bridges, cones and other stationary rigid bodies.

RMOD-K Tire 709 Tire Model

MotionSolve now supports RMOD-K Tire version 7.0.9 from T-Systems. This is a high fidelity tire model that can accurately predict tire response over a wide range of excitation frequencies and driving maneuvers. RMOD-K is typically used for durability, low frequency NVH and ride comfort simulations.

Depending on the scale of the road disturbances, two separate RMOD-K models are available to the user:

- Rigid Belt Model – adequate for excitation frequencies up to 100Hz and ground disturbance wavelengths greater than 100mm
- Flexible Belt Model – adequate for road excitations in excess of 100Hz and for very small ground disturbance wavelengths (down to millimeters). With flexible belt model, the user can select the number of nodes representing the mesh density of the tire carcass structure.

A third model suitable for misuse simulation is available on demand from T-Systems.

RMOD-K needs tire and road property files. The generation of tire property files requires a set of force/moment measurements as well as topology data, cleat measurements.

RMOD-K supports various road property files, including:

- C_ROAD analytical obstacles
- Regular Grid Roads
- Binary format road files representing digitized road surfaces.

RMOD-K is available on Windows platforms and requires a license. To obtain a license for RMOD-K tire, please contact admin@rmod-k.com or Altair.

Powertrain Model for Use in Vehicle Dynamics

The automotive extensions for MotionSolve now include an equation based powertrain model. This can be used with all MotionView/MotionSolve vehicle models for various vehicle dynamics and durability events. The underlying mathematics behind the powertrain is implemented as Control_State_Equation element (CSE). The equations represent the Internal Combustion Engine (IC Engine) and the clutch in the powertrain. Figure 3 below shows what the powertrain connects to and the signals it exchanges with the vehicle systems.

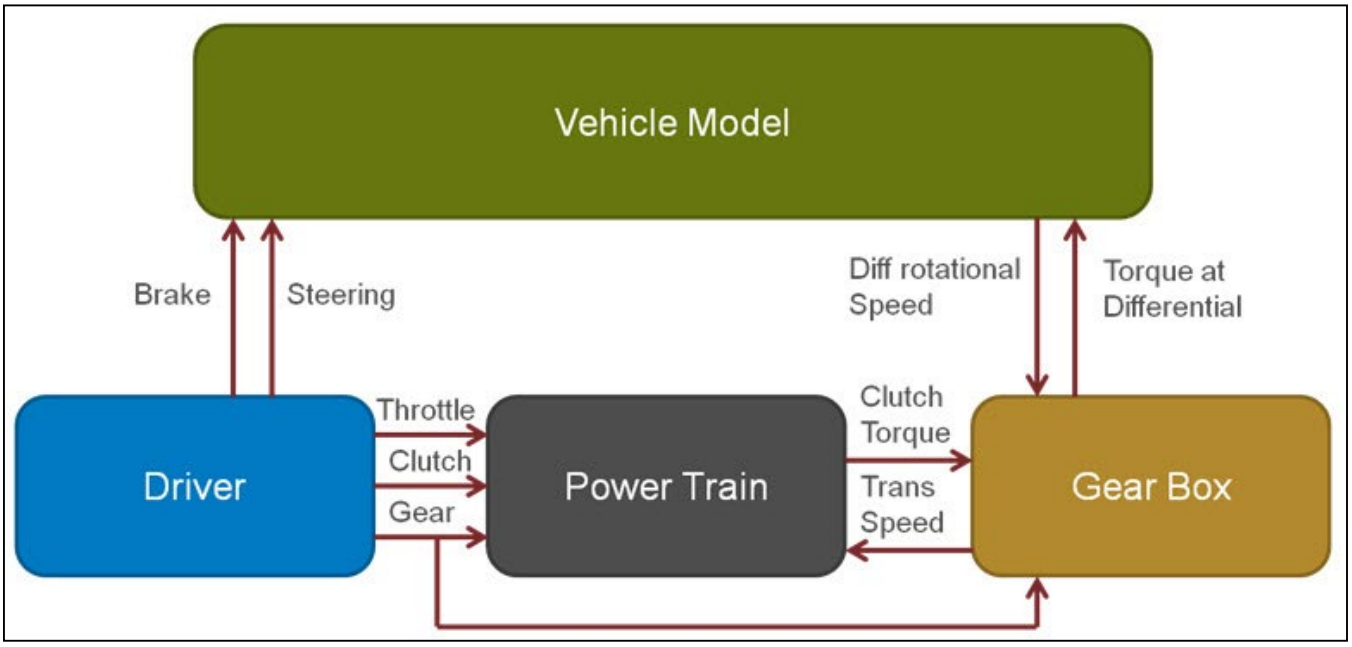


Figure 3: The Powertrain and how it interacts with the vehicle system

The model has the ability to read and interpolate into a 3-D Engine Torque Map to determine the torque output of an engine based on vehicle states and throttle input. It further models a clutch that transmits the torque to the gearbox and can be engaged and disengaged by a clutch input signal.

Static Design Factors Improvements

Static Design Factor (SDF) computations for suspensions may now be performed on models with multiple axles. Earlier releases of MotionSolve could handle only one axle. Now you can calculate the SDFs for any number of axles.

Tire Free Rolling (Angular) Speed

Calculations for the Free Rolling Omega, Ω_{free} have been revised. Now Ω_{free} for a tire is computed as:

$$\Omega_{free} = \frac{V_x}{R_e} = \frac{\Omega}{(1+\kappa)}$$

Where:

- V_x = Rolling Speed of the tire
- Ω = Omega Actual
- κ = Longitudinal Slip
- R_e = Tire Effective Rolling Radius

Tire Utilities

A handful of Tire States have been made available to Reference_Variable entities. The table below lists the function names in the mbdtire library, the required parameters and the description of the returned value.

Function Name	Parameter List	Description
TOTAL_PR	0 (no parameter)	Computes the total rolling resistance force developed by all the tires in the model
TIRE_RR_FORCE	Tire Force ID	Rolling resistance force as ratio between the torque and the rolling radius
TIREALPHA	Tire Force ID	Lateral Slip Angle
TIREKAPPA	Tire Force ID	Longitudinal Slip
TIREGAMMA	Tire Force ID	Tire inclination Angle

TNO MF and TNO SWIFT Tire Model Support

MotionSolve now supports the TNO MF and TNO SWIFT tire models. These add to the list of tires available via MotionSolve for vehicle dynamics, ride and durability analyses.

MF-Tyre is the Delft-Tyre standard implementation of the renowned Pacejka Magic Formula that includes the latest developments. With MF-Tyre the engineer can simulate validated steady state and transient behavior up to 8Hz, making it the ideal tire model for handling and control prototyping analyses. Pure cornering and braking, as well as combined conditions are accurately modeled. For more information on MF-Tyre, please visit http://www.tno.nl.

MF-Swift is the high frequency (100Hz) extension to MF-Tyre. MF-Swift adds generic 3D obstacle enveloping and tire belt dynamics to MF-Tyre's tire-road contact force and moment calculations. Typical application areas for MF-Swift are vehicle comfort analysis, suspension vibration analysis, development of vehicle control systems, such as ABS or ESP, and handling and stability analysis, like braking and power-off in a turn on an uneven road. For more information on MF-Swift, please visit http://www.tno.nl.

To obtain a license for TNO SWIFT Tire, please contact tno@tno.com or Altair. Please note that MF-Tyre does not require a license.

Back to top

Software Behavior Changes

User Subroutine Search Behavior

This capability is useful for debugging large models with many DLLs. MotionSolve prints out the locations (absolute paths) where it searches for user subroutines that are referenced in the input deck.

Warning for Special Characters

While converting *.adm* decks to *.xml*, unwanted warnings were issued if the labels/comments contained special characters (& and `). These have been removed.

Back to top

ADAMS Compatibility

MotionSolve is highly compatible to ADAMS. It can read in input ADM and ACF format. It is even binary compatible as far as the user subroutine DLLs are concerned. This section describes in detail the compatibility between MotionSolve and ADAMS. The tables below have the following key:

- Red: An Adams capability is not available in MotionSolve. When not applicable, it is stated explicitly
- Blue: A MotionSolve capability is not available in Adams

ADM Compatibility

For most ADAMS modeling elements there are equivalent MotionSolve elements. In some cases, a combination of MotionSolve elements may be used to provide equivalent functionality. The ADM translator in MotionSolve addresses this issue.

ADAMS	MotionSolve
2d PART	Subsystem_Planar
ACCGRAV	Force_Gravity
ARRAY	Reference_Array
BEAM	Force_Beam
BUSHING	Force_Bushing
CONTACT	Force_Contact
COUPLER	Constraint_Coupler
CURVE	Reference_ParamCurve
CVCV	Constraint_CVCV
DEBUG	Debug_Output
DIFF	Control_Diff
END	Not applicable
EQUILIBRIUM	Param_Static
FEMDATA	Not applicable

ADAMS	MotionSolve
MOTION	Motion_{Joint, Marker}
MREQUEST	Not applicable
NFORCE	Force_MultiPoint
Not Supported	Constraint_PTdCV
Not Supported	Constraint_PTd/SF
Not Supported	Constraint_PTSF
Not Supported	Constraint_CVSF
Not Supported	Constraint_SFSF
Not Supported	Force_PTdSF
Not Supported	Reference_DeformCurve
Not Supported	Reference_DeformSurface
OUTPUT	Not applicable
PART	Body_Rigid
PINPUT	Control_PlantInput

FIELD	Force_field
FLEX_BODY	Body_Flexible, Reference_FlexData
FRICTION	Force_JointFriction
GCON	Constraint_Mate
GEAR	Constraint_GEAR
GFORCE	Force_Vector_{OneBody, TwoBody}
GRAPHICS	Post_Graphics
GSE	Control_StateEqn (type: Usersub)
IC	IC
INTEGRATOR	Param_Transient
JOINT	Constraint_Joint
JPRIM	Constraint_Jprim
KINEMATICS	Not applicable
LIST/NOLIST	Not applicable
LSE	Control_StateEqn (type: Linear)
MARKER	Reference_Marker
MATRIX	Reference_Matrix
MFORCE	Force_FlexModal

POINT_MASS	Body_Point
POUTPUT	Control_PlantOutput
PREFERENCES	Not applicable
PTCV	Constraint_PTCV
REQUEST	Post_Request
RESULTS	Not applicable
SENSOR	Sensor_{Event,Evaluate}
SFORCE	Force_Scalar_ {OneBody,TwoBody}
SPLINE	Reference_Spline
SPRINGDAMPER	Force_SpringDamper
STRING	Reference_String
SURFACE	Reference_ParamSurface
TFSISO	Control_SISO
UCON	Constraint_UserConstr
UNITS	Param_Unit
VARIABLE	Reference_Variable
VFORCE	Force_Vector_ {OneBody,TwoBody}
VTORQUE	Force_Vector_ {OneBody,TwoBody}

ACF Compatiblity

For most ADAMS commands there are equivalent MotionSolve commands. In some cases, a combination of MotionSolve commands may be used to provide equivalent functionality. The ACF translator in MotionSolve addresses this issue.

ADAMS	MotionSolve
ACCGRAV	Force_Gravity
ACTIVATE	Activate
ARRAY	Reference_Array

ADAMS	MotionSolve
MENU	Not applicable
MOTION	Motion_{Joint,Marker}
OUTPUT	Not applicable

BEAM	Force_Beam
BUSHING	Force_Bushing
CONTROL	UserProgramControl
DEACTIVATE	Deactivate
DEBUG	DebugOutput
EQUILIBRIUM	Param_Static
FIELD	Force_field
FILE	Load_{Model,Command}
FLEX_BODY	Body_Flexible
FRICTION	Force_JointFriction
GCON	Constraint_Mate
GFORCE	Force_Vector_TwoBody
H3D	H3Doutput
IC	IC
INFO	Not applicable
INTEGRATOR	Param_Transient
JOINT	Constraint_Joint
KINEMATICS	Not applicable
LINEAR	Param_Linear
MARKER	Reference_Marker

PART	Body_Rigid
POINT_MASS	Body_Point
PREFERENCES	Not applicable
RELOAD	Load_{Model,Command}
REQUEST	Post_Request
SAVE	Save
SENSOR	Sensor_{Event,Evaluate}
SFORCE	Force_Scalar_TwoBody
SHOW	Not applicable
SIMULATE	Simulate
SPLINE	Reference_Spline
SPRINGDAMPER	Force_SpringDamper
STOP	Stop
STRING	Reference_String
TIME	Not applicable
UCON	Constraint_UserConstr
UNITS	Param_Unit
VARIABLE	Reference_Variable
VFORCE	Force_Vector_TwoBody
VTORQUE	Force_Vector_TwoBody

User Subroutine Compatibility

For almost all ADAMS user subroutines there are equivalent MotionSolve subroutines. The calling sequences and parameter types are identical. This enables the DLLs to be binary compatible, i.e. a DLL created for ADAMS will run directly in MotionSolve and vice versa.

ADAMS	MotionSolve
CFFSUB	CFFSUB
CNFSUB	CNFSUB
CONSUB	CONSUB

ADAMS	MotionSolve
GETINT	GETINT
GETMOD	GETMOD
GETSLV	GETSLV

COUSUB, COUXX, COUXX2	COUSUB, COUXX, COUXX2
CURSUB	CURSUB
DIFSUB	DIFSUB
DMPSUB	DMPSUB
FIESUB	FIESUB
GFOSUB	GROSUB
GSE{SUB, XU, XX, YU, YX}	GSE{SUB, XU, XX, YU, YX}
GSE_{DERIV, UPDATE}	Not Supported
GSE_ {OUTPUT, SAMP}	Not Supported
MFOSUB	MFOSUB
MOTSUB	MOTSUB
RELSUB	RELSUB
REQSUB	REQSUB
SAVSUB	SAVSUB
SENSUB	SENSUB
SEVSUB	SEVSUB
SFOSUB	SFOSUB
SPLINE_READ	SPLINE_READ
SURSUB	SURSUB
TIRSUB	Not Supported
UCOSUB	UCOSUB
VARSUB	VARSUB
VFOSUB	VFOSUB

GETSTM	GETSTM
GET_SPRASE_ MATRIX_DATA	GET_SPRASE_MATRIX_ DATA
GETVER	GETVER
GTARAY	GTARAY
GTCMAT	GTCMAT
GTCURV	GTCURV
GTINAM	GTINAM
GTONAM	GTONAM
GTSTRG	GTCMAT
GTUNTS	GTUNTS
GET_MATRIX_ INFO	GET_MATRIX_INFO
HAVSIN	HAVSIN
IMPACT	IMPACT
ISTRNG	ISTRNG
MODFNC	MODFNC
MODINF	MODINF
NMODES	NMODES
POLY	POLY
PUT_SPLINE	PUT_SPLINE
QUISPL (Not in ADAMS)	QUISPL
RCNVRT	RCNVRT
RELPAR	RELPAR

VTOSUB	VTOSUB
NALYS	ANALYS
DATOUT	DATOUT
MODIFY	MODIFY
ADD_MASS_ PROPERTY	ADD_MASS_PROPERTY
BODY_MASS_ PROPERTY	BODY_MASS_PROPERTY
AKISPL	AKISPL
BISTOP	BISTOP
CHEBY	CHEBY
CUBSPL	CUBSPL
ERRMES	ERRMES
FORCOS	FORCOS
FORSIN	FORSIN
GETCPU	GETCPU
GETINM	GETINM
GET_FULL_ MATRIX_DATA	GET_FULL_MATRIX_DATA
GET_GRAVITY	GET_GRAVITY

RSTRNG	RSTRNG
SAVPAR	SAVPAR
SHF	SHF
STEP	STEP
STEP5	STEP5
STR2INTARY	STR2INTARY
STR2DBLARY	STR2DBLARY
SUBTRACT_ MASS_ PROPERTY	SUBTRACT_MASS_ PROPERTY
SYSARY	SYSARY
SYSFNC	SYSFNC
SYSPAR	Not Supported
TCNVRT	TCNVRT
TIMGET	TIMGET
TIRARY	Not Supported
Not Supported	UCOMAR
UCOVAR	UCOVAR
USRMES	USRMES

Back to top

Resolved Issues

ADM File Import

The ADAMS GSE statement attributes of xxflag, xuflag, yxflag, and yuflag are now correctly supported when reading an `.adm` file.

Contact Force Graphics

Models containing one or more contact entities could result in the inability to view contact force graphics in HyperView. This has been corrected.

Deformable Surface

The Deformable Surface entity contains a "flip_normal" attribute. The normal for the surface was calculated assuming no penetration to the surface at t=0. This assumption was not always correct and the calculation has been corrected.

Flex-bodies

Stress/strain results for flex-bodies issue related to unmatched units between RADIOSS and MotionSolve has been resolved.

If the time unit used in the MotionSolve input deck was not seconds, the model may have incorrect transient behavior. This has been corrected.

H3D Animation Speed In 11.0

H3D animation performance issues, seen with 11.0 in HyperView for results containing flex-bodies have been resolved.

HTire Tire Model Licensing

The HTire tire model was checking for an FTire license. This has been resolved.

Initial Velocity Calculations

If joint initial velocity and a motion are applied on the same joint, there could be an inconsistency in the initial velocity computation. This has been corrected.

Motion Expression

If a motion is a function of a state dependent variable in the case where multiple simulations are run via a single set of commands, the solver run could fail. This issue is fixed.

Offset issues when Flexible Marker location & node position are not coincident

While Flexible Markers are typically coincident with a node in the finite element mesh, this is not a requirement. Flexible Markers can have a non-zero offset from a node.

Prior to 12.0, MotionSolve would detect this scenario, reposition the Flexible Marker so that it was coincident with its associated node and continue. User data would be ignored. This issue is fixed now. Now the Flexible Marker will remain at the location specified in the Flexible Marker definition and will rigidly move with respect to its associated node.

Parametric Curve

In the case where the initial conditions for a parametric curve were out of range, the simulation may fail to start. This has been fixed.

Post_Request Initialization

In the case that a Post_Request cannot be initialized properly, the solver would perform a solution and quit prior to generating the results. Now it detects the error and stops.

Python Splines

AKISPL, CUBSPL and BISTOP expressions inside Python user subroutines would return a type of vector instead of scalar; thus, making these expressions unusable. This has been fixed.

Run Termination

If a MotionSolve run initiated by MotionView were stopped, the MotionSolve process would continue being active, possibly corrupting the result files. This issue has been corrected.

Unexpected Euler Angle Rotations for Markers During Static Equilibrium

Marker displacement REQUESTS would report incorrect values for orientations, when expressed as B-313 Euler Angles during static equilibrium. This error occurs only when the FIM_D (Force Imbalance Method—Dynamics) analysis option was used.

The issue was traced to numerical truncation associated with the Marker transformation matrix—the transformation matrix had minor errors in it and it was not exactly orthonormal. Re-orthonormalization of the transformation matrix prior to reporting the Euler Angles fixed this issue.

Fixed error in angles reported in Marker Displacement request during FIM_D.

SAVE/RELOAD: Inconsistent Velocities After a SAVE/RELOAD Operation

Models that use the Save and Load_Model statements to carry out multiple simulations displayed inconsistent velocities and omegas between two consecutive simulations. This issue affected only fast rotating bodies that use a slightly different formulation than "regular" bodies. This issue has been fixed.

SAVE/RELOAD: Output Not Generated for RELOAD Operations

In earlier versions, MotionSolve did not generate ASCII results files (.plt) for models that use the Save and Load_Model statements to run multiple simulations. This has been fixed. Output files are now generated for every Load_Model invocation.

SAVE/RELOAD: Unexpected Orientations After a SAVE/RELOAD Operation

In some scenarios, the Euler Angles associated with the Part Principle Coordinates System would mysteriously change after a model was SAVED and immediately RELOADED. This behavior was traced to back to truncation of inertia data during the SAVE operation. Inertia data was saved to only six significant figures.

In 12.0, inertia data is saved to twelve significant figures. With this change, a SAVE followed by a RELOAD does not affect the orientation of the Part Principle Coordinates System.

Please note this is only an output reporting issue, as the concept of a Part Principal Coordinate System is not used in MotionSolve.

Support for Multiple PINPUT and POUTPUT in the XML File

Earlier versions of MotionSolve worked with just one Plant_Input and one Plant_Output. When many Plant_Input and Plant_Output statements were provided, MotionSolve would just use the first.

The Linear Analysis capability in 12.0 has been enhanced as follows:

- New language has been added to the SIMULATE command for the ANALYSIS=LINEAR option. This analysis requires a Plant_Input and a Plant_Output to be specified.
- The specified inputs and outputs are used to calculate the state (ABCD) matrices.

Units Support For Damping In Bushing Property Files

Automotive Bushing (field) based on property file now applies unit conversion factors to the rotational damping terms. Previously, the property file units were ignored.

Back to top

Highlights of the RADIOSS 12.0 Release

Major enhancements for Stress and Strength, Noise and Vibrations, Powertrain Durability, and Heat Transfer Analysis (bulk data format).

- AMSES Large Scale Eigenvalue Solver
- Fast Parallel Solver for Modal Frequency Response
- Modeling the Effect of Heavy Incompressible Fluids on Structural Vibrations
- Equivalent Radiated Power and Far Field Sound Level Pressure
- One Step Transfer Path Analysis
- Heat Transfer Analysis based on Contact Distance
- Transient Heat Transfer Analysis
- Axisymmetric Analysis
- Improved Convergence of Contact Analysis with Friction
- Contact-friendly Second Order Solid Elements
- Modeling of Gaskets
- Nonlinear Bushing Elements
- 1D and 3D Bolt Pretension

Major enhancements for Crash & Safety, Blast and Forming (block format).

- Parallel Performance
- Starter SMP Parallelization
- SPH INLET & OUTLET Parallelization
- Tied Contact with Rupture
- Crushable Beam
- Accurate Airbag Simulation with the FV Method
- New Joint Formulations
- Transformations and Rotations of Submodels
- Model Sensitivity Control

Contents

12.0 - RADIOSS for Stiffness, Strength and Stability (Bulk Data Format)

- Axisymmetric Analysis
- Improved Convergence of Contact Analysis with Friction
- Contact-friendly Second Order Solid Elements
- Modeling of Gaskets
- Nonlinear Bushing Elements

1D and 3D Bolt Pretension

GPU Support for Faster Static Analysis

Heat Transfer Analysis Based on Contact Distance

Transient Heat Transfer Analysis

Gasket Elements

Pretension Analysis of 3D Bolts

GPU Can be Used to Speed up Static Analysis

Axisymmetric Analysis

CFAST and PFAST are Now Available for Modeling Fasteners

The Initial Clearance Between the Contact Surfaces Can be Prescribed

Nonlinear Springs

SUBCASE Specific Loads from DMIG Data

Plasticity with Kinematic and/or Mixed Hardening Now Available for Nonlinear Analysis

Boundary Conditions Can Change for Continuation Subcases

Internal (MPCF) and External (OLOAD) Forces Written to the .op2, .pch, and .h3d Files for Nonlinear Analysis

Support of Elasto-plastic Orthotropic Material with TSAI-WU and CRASURVT Yield Criteria for Composite Shell Materials

Mechanical Strain Written to the HyperMesh .res File

Bolt Pretension

Use of SUBCOM and SUBSEQ to Combine Static Analysis SUBCASE Results

Modal STRESS and DISPLACEMENT for All Modes and Residual Vectors Written to the .op2 and .pch Files

Modeling and Analysis of Seam Welds

GPFORCE Output Available for Eigenvalue Analysis

CSTRESS and CSTRAIN Output Available for Eigenvalue Analysis

CSTRESS, CSTRAIN and CFAILURE Output Available for Transient Analysis

CSTRESS and CSTRAIN Output Available for Frequency Response Analysis

Output of Modal Grid Point (GPSTRESS) and Element Corner Stresses for the CMS Modes to the .op2 File

Ply-based Composite Definition Can Now be Used in Nonlinear Analysis

SMP and SPMD Parallel Analysis Now Available for Nonlinear Analysis

Plasticity with Kinematic and/or Mixed Hardening Now Available for Solid Elements

RBAR Data Supported in Nonlinear Analysis

Rigid Element Force (MPCF) Results Written to the .op2 File for Nonlinear Analysis

Pre-stressed Buckling Analysis

Stress and Strain Recovery Coefficients Available for CBUSH

ASCII Results are Now Available for Stress and Strain Results in NLGEOM, IMPDYN and EXPDYN Analysis

Failure Theories Can Now be Specified on PCOMPP and PCOMPG Data for NLGEOM, IMPDYN and EXPDYN Analysis

CROD Stress Can Now be Recovered in NLGEOM, IMPDYN and EXPDYN Analysis

Fatigue Damage and Life Results are Now Written to the HyperMesh .res Results File

MPCFORCES Output Written to the .h3d File for Static Analysis

Unreferenced SET Data Written to the .op2 File

SET Data Written to the .h3d File

Negative Values for LDM and Scale Factors Now Allowed in FATLOAD Data

Drilling Stiffness of Shells Can be Turned Off for Nonlinear Analysis

Composite Laminate Added to Account for Offset Elements

Composite Failure Index and Strength Ratio Now Output to the .op2 File

Geometric Nonlinear Solutions

Default Projections and Checks for CWELD Elements

New Parameters

New Subcase Information Entries

New Bulk Data Entries

Enhanced Subcase Information Entries

Enhanced Bulk Data Entries

Resolved Issues

12.0 - RADIOSS for Noise and Vibrations (Bulk Data Format)

AMSES Large Scale Eigenvalue Solver

Fast Parallel Solver for Modal Frequency Response

Modeling the Effect of Heavy Incompressible Fluids on Structural Vibrations

Equivalent Radiated Power and Far Field Sound Level Pressure

One Step Transfer Path Analysis

MFLUID is Available for Modeling a Virtual Fluid

Automatic One Step Transfer Path Analysis

Added PARAM,AMSESLM So the Large Mass Method can be Used with EIGRA

Added PARAM,GE_MOD to Override All Material, Element, and Property GE Data

Equivalent Radiated Power (ERP)

Component Dynamic Synthesis (CDS) Super Elements

Output of Applied Loads for Frequency Response and Transient Analysis

Ability to Control the Output of Results from Interior Points and Elements in Super Elements

Automated Multi-level Sub-structuring Eigenvalue Solver

Hybrid Damping for Direct Dynamic Analysis

Frequency Dependent Spring Property

Speed of the Modal Frequency Response Analysis Solution

Results for a SET of GRID can be Requested from a CMS Super Element

The Fluid-Structure for Coupling Matrix Can Now be Written to the .pch File

SPCD Data Created for CMS Super Elements for Each Mode During Eigenvalue Analysis

PFMODE and PFPANEL Can Write Results to the .h3d, .pch or Both Files

PFMODE Output Can Now be Determined by Displacement, Velocity or Acceleration

RMS Results Only Output Option Added

Support of Additional Damping in the Interface to FastFRS

Support of Acoustic Loading in the Interface to FastFRS

PSDF Output to .h3d File Expanded

PSDF Output to .op2 File Expanded

PSDF Plotting Output Expanded

CMDE Output Now Accounts for Both Viscous and Structural Damping

Specifications of the Matrices Used from H3D DMIG File

Improved Control of ERP Calculations

ACMODL Statistics Written to the .out File

EIGRA Solution Time Reduced for Large Eigenvalue Problems

EIGRA Can be Used to Solve Larger Problems

EIGRA and AMLS Can Now be Used for Pre-stressed Normal Modes Analysis

Modal Frequency Response Solution Time Reduced for Problems Without Structural or Viscous Damping

AVL/EXCITE Interface

SIMPACK Interface

Allow Craig-Chang Super Elements in Residual Run Using AMLS/AMSES

Time Varying W3 and W4 for Transient Response Analysis

Static Loads Can Now be Applied Dynamically for Transient Analysis

Modal Participation Ratio Output

AUTOSPC for AMLS Solver Interface

SET Data Specifications for Rigid Element Output on the MODEL Data

Modal Stress Data Can be Exported Using PARAM,LMSOUT

Element FORCE Results are Available for CBUSH, CELASi, CDAMP, and CVISC Elements in Transient Response Analysis

Element FORCE Results are Available for CBUSH, CELASi, CDAMP, and CVISC Elements in Frequency Response Analysis

Element FORCE Results are Output From Eigenvalue Analysis

Element Strain Energy (ESE) is Now Written to the .op2 File

Element Strain Energy (ESE) for Frequency Response is Now Written to the .pch File

Element Energies (ESE/EKE/EDE) for DMIG (CMS SE) Elements are Now Written to the .out File

MPCFORCE Output is Now Available from Modal Frequency Response Analysis

A File to Allow Optimization of CMS Super Elements Can be Generated

Sets of RIGID Elements Can Now be Specified with the Bulk Data SET Data

New Parameters

New Subcase Information Entries

Enhanced Subcase Information Entries

New I/O Option Entries

New Bulk Data Entries

Enhanced I/O Option Entries

Enhanced Solution Control Entries

Enhanced Bulk Data Entries

Resolved Issues

12.0 - RADIOSS for Kinematic Analysis (Bulk Data Format)

Ply-based Composite Definition Can Now be Used in MBD Analysis

CB and CC Flexbody Generation Time Reduced

Pre-loaded CB and CC Flexbodies Can be Generated

Residual Vectors Can be Used to Increase the Accuracy of CB and CC Flexbodies

CB and CC Flexbodies Can Contain Reduced Loads

CMS Flexbodies Can Now be Created Based on PUNCH DMIG Input Data

1D Element Forces Included in Flexbody Output

Enhanced Bulk Data Entries

12.0 - RADIOSS - General (Bulk Data Format)

GRID Data Replication

AUTOSPC DOF Written as SPC/SPC1 to a PUNCH File

The SYSSETTING Command SAVEFILE Has Been Added to Save Analysis Results from Previous Runs

-compress Option Now Works with a Tolerance

Solution Control AUTOSPC Command

GENEL Element

No Available License, No Worries

Bulk Data SET1 and SET3 Formats Supported

Passing of Metadata from the Input File to a Resulting XML File

New 3D Orthotropic Material Definition (MAT9ORT)

Automatic Input Update

Long-field Free Format Input

Compressed Input Files

Contributions from Individual MPC and Rigid Elements are Listed in the GPFORCE Output

DMIG Mass Effect is Included in the Generation of RFORCE and GRAV Loads

Warning Messages Can be Turned Off

Shell Element Offset Can be Specified as Top or Bottom

Stress and Strain Results for CBAR, CBEAM and CROD are Now Written to the .pch and .op2 Files

SPCFORCE Output is Now Available for NLGEOM, IMPDYN and EXPDYN Analysis

SPCFORCE Output is Now Available for Eigenvalue Analysis

MAT9ORT Can Now be Used in NLGEOM, IMPDYN and EXPDYN Analysis

PARAM,ALPHA1 and PARAM,ALPHA2 Can Now be Used in NLGEOM, IMPDYN and EXPDYN Analysis

Multiple XDAMP Data Can be Used in EXPDYN Analysis

Shell Elements have Drilling Degree of Freedom Stiffness in NLGEOM and IMPDYN Analysis

The Shell Element Thickness is Now Stored with the Model Information in the .h3d File

Structural Damping and a Stress Recovery Coefficient Can Now be Specified on CELAS4 Data

AMSES Statistics Written to the .stat File

Mass Properties Written H3D CMS Super Element Files

Rigid Elements are Now Included in the .mnf File

Support of Problems with Many Constraint Equations Due to Large RBE3 Elements

Scratch Files Created by FastFRS are Now Deleted at the End of the Run

Disk Space Estimate Improved for Modal Analysis

Support of Platform MPI Version 8 on Windows

Element Types, Properties, and Materials Can Now be Used to Define Sets of Grids Using Bulk Data SET

RIGID Option for Stiffness Entry on Bushing Property Definition

Improved SMP Parallelization for Iterative PCG Solver

Solver Run Manager GUI

PBEAML Can be Used with HyperBeam

Duplicate Property and Material Data is Allowed

New Parameters

New SYSSETTINGS Entries

New I/O Option Entries

New Solution Control Entries

New Subcase Information Entries

New Bulk Data Entries

Enhanced PARAM Entries

Enhanced I/O Option Entries

Enhanced Solution Control Entries

Enhanced Bulk Data Entries

Resolved Issues

12.0 - RADIOSS for Crash and Safety (Block Format)

Tied Contact with Penalty Formulation (Spotflag=25)

Tied Contact with Rupture (Spotflag from 20 to 22)

HEPH Improved Hourglass Control

Joints

Crushable Beam (/PROP/TYPE44)

Springs and Beams

Transformations and Rotations of Submodels

Airbags

12.0 - RADIOSS for Forming (Block Format)

AMS Compatibility with Mesh Adaptivity

Contact TYPE21

Material LAW78

Initialization of Stress/Strains (Reading & Output) for Brick Elements

Initialization of Nodal Temperature (Reading & Output)

12.0 - RADIOSS for Blast Simulation (Block Format)

Material LAW5

Material LAW51

Material LAW79

/TH/BRIC/BFRAC

12.0 - RADIOSS - General (Block Format)

Time Step Control for Contact Interfaces (Advanced Mass Scaling)

/INTER/TYPE7

/INTER/TYPE7, TYPE11 and TYPE19

/INTER/TYPE7, TYPE10, TYPE11, TYPE19 and TYPE20

Material LAW2 (Johnson-Cook)

Material LAW25 (Composite)

Solid Elements

Pressure Loads

Centrifugal Loads

/INIVEL/AXIS (Starter)

/INIV/AXIS (Engine)

Cylindrical Coordinate System

Input Parametrization

Model Sensitivity (/DAMP/INTER)

Single Precision

Starter Parallelization (SMP)

Engine Parallel Performance (SPMD)

SPH INLET & OUTLET Parallelization (SPMD)

GPU Support for Implicit Iterative Solver (Hybrid Intel MPI)

Anim Output
/ANIM/NODA/DAMA2
Time History Output
ABF (Altair Binary Format) Output
Engine Messages

Introduction

RADIOSS is a state-of-the-art finite element solver uniting implicit and explicit integration schemes for the solution of a wide variety of engineering problems, from linear statics and linear dynamics to complex nonlinear transient dynamics and mechanical systems. This robust, multidisciplinary solver enables designers to maximize performance related to durability, NVH, crash, safety, manufacturability, and fluid-structure interaction, in order to bring innovative products to market faster.

Back to top

RADIOSS for Stiffness, Strength, Stability (Bulk Data Format)

Axisymmetric Analysis

Axisymmetric structures can now be modeled with 3 and 6 grid triangular elements. In additional to gravity, rotational force, line, and thermal loads, axisymmetric pressure loads can be applied to the structure. Axisymmetric orthotropic material properties are available.

Improved Convergence of Contact Analysis with Friction

The convergence rate for nonlinear analysis involving contact with friction has been greatly improved through an improved contact algorithm and the use of a non-symmetric equation solver. For high values of friction it is necessary to include non-symmetric terms in the stiffness matrix. The non-symmetric equation solver allows these matrices to be used, which leads to a significant increase in the convergence rate for problems with high friction coefficients.

Contact-friendly Second Order Solid Elements

Traditional second order solid elements often cause difficulties in solution of contact problems. They exhibit zero or even negative nodal forces at corner nodes, which can spoil the convergence of nonlinear process and produce very uneven distribution of contact pressure. To remedy this, special versions of second order elements have been implemented – these elements produce greatly improved results with contact while retaining the benefits of high accuracy of second-order elements.

Modeling of Gaskets

The effect of gaskets on bolted connections can now be modeled. 6, 8, 12, and 16 grid gasket elements are now available to model the contact and gasket compression between bolted parts. Nonlinear material properties and friction are taken into account.

Nonlinear Bushing Elements

Nonlinear bushing elements have been implemented to facilitate realistic simulation of bushings that use complex internal structure and nonlinear materials. Both the full 3D and spring-like 1D versions of nonlinear bushings are available.

1D and 3D Bolt Pretension

Pretension can be applied to bolts modeled with 1D or 3D elements. An easy to use method has been developed to specify the bolt pretension for bolts modeled with 3D elements. The pretension supports the specification of pretension forces and/or adjustment (of relative displacements) in nearly arbitrary combinations and sequences.

GPU Support for Faster Static Analysis

RADIOSS can now use the GPU to speed up linear analysis on Linux systems.

Heat Transfer Analysis Based on Contact Distance

Thermal conductivity based on the distance between parts can now be modeled. This allows for heat transfer between parts that are about to come into contact and also higher conduction rates when the parts do come into contact.

Transient Heat Transfer Analysis

Time varying heat transfer analysis can now be preformed. Time varying thermal loads (flux and heat generation) are allowed, as well as time varying temperature boundary conditions for both the structure and the ambient fluid.

Gasket Elements

Gasket elements are now available for modeling gaskets in nonlinear analysis. The CGASK6, CGASK8, CGASK12, and CGASK16 elements are used to define gasket elements with 6, 8, 12, and 16 grids respectively. The PGASK and MGASK data are used to define the gasket properties and material.

Pretension Analysis of 3D Bolts

Bolts modeled with solid elements can now have pretension defined using the PRETENS data.

GPU Can be Used to Speed Up Static Analysis

Graphic cards can be used to accelerate static analysis by offloading most of the computation intensive work to GPU, while overlapping the communication and data transfer between CPU cores and GPU. The speedup in equation solver can be up to 4X, and the overall by up to 3X compared to Quad-core Intel Xeon run. This is available on Linux only. The Command Line Option "-GPU" is used to activate the GPU computing. Currently, only one graphic card is supported, and thus "–GPUID" can be used to pick the desired graphic card for computation when multiple are present. NVIDIA Fermi and Kepler architecture based Tesla and Quadro graphic cards are supported. Tesla C2050/C2070/M2090/K10/K20, Quadro 6000/K5000/K6000 cards are recommended for computing by NVIDIA. The proper driver for graphic card has to be installed from the user end before launching with option "-GPU".

Axisymmetric Analysis

Axisymmetric Analysis can now be performed using the CTRIAX6 and CTAXI elements, PAXI element property, MAT1, MATT1, MAT3 and MATT3 material properties, and PLOADX1 pressure loading.

CFAST and PFAST are Now Available for Modeling Fasteners

The fastener connects two surface patches defined shell elements or properties (PSHELL). The fastener has a diameter and stiffness that can be defined in a local coordinate system.

The Initial Clearance Between the Contact Surfaces Can be Prescribed

There is now an item in the PCONT data to specify an initial clearance between the master and slave contact surfaces. The does not depend on the actual distance between the surfaces.

Nonlinear Springs

Nonlinear force vs. deflection curves can now be defined for CBUSH elements using the KN TYPE in the PBUSHT data.

SUBCASE Specific Loads from DMIG Data

A specific loading vector can be pulled from DMIG data and applied as a load in a single SUBCASE using the P2GSUB data. For example, to apply the second load vector of DMIG PAX in a specific SUBCASE, use P2GSUB=PAX(2) data in that SUBCASE.

Plasticity with Kinematic and/or Mixed Hardening Now Available for Nonlinear Analysis

The MATS1 data now allows specification of Kinematic and/or Mixed Hardening rules through expanded options for the HR data.

Boundary Conditions Can Change for Continuation Subcases

The SPC set can change from the previous subcase when the continuation subcase (CNTNLSUB) is used for nonlinear analysis. This allows for previously SPC'ed DOF to be released.

Internal (MPCF) and External (OLOAD) Forces Written to the .op2, .pch, and .h3d Files for Nonlinear Analysis

MPCF and OLOAD analysis results are now written to the `.op2`, `.pch`, and `.h3d` files for NLGEOM, IMPDYN, and EXPDYN nonlinear analyses.

Support of Elasto-plastic Orthotropic Material with TSAI-WU and CRASURVT Yield Criteria for Composite Shell Materials

MATX25 material definition has been added for NLGEOM, IMPDYN, and EXPDYN nonlinear analyses.

Mechanical Strain Written to the HyperMesh .res File

Under thermal analysis, the separate mechanical and thermal strain components are written to the HyperMesh `.res` file.

Bolt Pretension

Pretensioned bolts can be modeled with 1D elements. The pretension analysis can be applied to linear and nonlinear static subcases. These static subcases can then be used in pre-loaded dynamic analysis.

Use of SUBCOM and SUBSEQ to Combine Static Analysis SUBCASE Results

The new SUBCOM data can be used to generate results that are a linear combination of preceding linear and nonlinear static SUBCASE results. The new SUBSEQ data is used to specify the weighting factors.

Modal STRESS and DISPLACEMENT for All Modes and Residual Vectors Written to the .op2 and .pch Files

Using the keywords STRESS(MODAL) and DISPLACEMENT(MODAL) in an eigenvalue, modal FRF, or modal transient analysis SUBCASE will cause the modal stresses and displacements to be output. These modal results include both the structural modes and the residual vector modes. The `.op2` or `.pch` file generation can be specified using the OUTPUT command (OUTPUT=OUTPUT2 or OUTPUT=PUNCH) or in the result specification: STRESS(MODAL,OP2), DISP(MODAL,PUNCH), etc.

These results can be used when third party fatigue codes combine the stress modes using modal participation coefficients from a transient analysis to calculate transient stresses.

Modeling and Analysis of Seam Welds

CSEAM and PSEAM data have been added for the modeling and analysis of seam welds.

GPFORCE Output Available for Eigenvalue Analysis

Grid Point Force Balance information is now available for structural modes in `.h3d`, `.op2`, and `.pch` results files.

These results can be used when third party fatigue codes combine the modal forces using modal participation coefficients from a transient analysis to calculate transient forces in welded, riveted, or bolted connections.

CSTRESS and CSTRAIN Output Available for Eigenvalue Analysis

Composite stress and strain results are now available for structural modes in the `.h3d` result file.

CSTRESS, CSTRAIN, and CFAILURE Output Available for Transient Analysis

Composite stress, strain, and failure index results are now available for transient analysis in the `.h3d` result file.

CSTRESS and CSTRAIN Output Available for Frequency Response Analysis

Composite stress and strain results are now available for frequency response analysis in the `.h3d` result file.

Output of Modal Grid Point (GPSTRESS) and Element Corner Stresses for the CMS Modes to the .op2 File

During the CMS Structural Super Element generation, the CMS modal stresses can be written to the `.op2` file. These include the element centriodal and corner stresses for plates and solids, as well as the grid point stresses for grids of a solid mesh.

Ply-based Composite Definition Can Now be Used in Nonlinear Analysis

PCOMPP ply based composite property data can now be used in the NLGEOM, IMPDYN, and EXPDYN nonlinear analyses.

SMP and SPMD Parallel Analysis Now Available for Nonlinear Analysis

The SPMD parallel option can now be used for analysis types NLGEOM, IMPDYN, and EXPDYN through the use of the `-rnp` and `-rnt` command line options. `-rnt` specifies the number of SMP parallel threads, and `-rnp` specifies the number of SPMD parallel processors.

Plasticity with Kinematic and/or Mixed Hardening Now Available for Solid Elements

The MATS1 data now allows specification of Kinematic and/or Mixed Hardening rules through expanded option for the HR data.

RBAR Data Supported in Nonlinear Analysis

RBAR rigid bar data can now be used in the NLGEOM, IMPDYN, and EXPDYN nonlinear analyses.

Rigid Element Force (MPCF) Results Written to the .op2 File for Nonlinear Analysis

MPCF analysis results are now written to the `.op2` file for RBE2, RBE3, and RBAR elements used in NLGEOM, IMPDYN, and EXPDYN nonlinear analyses.

Pre-stressed Buckling Analysis

It is now possible to perform a linear buckling analysis of a structure in a pre-stressed state.

STATSUB(PRELOAD) is used to identify the pre-stress subcase.

STATSUB(BUCKLING) is used to identify the loading subcase.

Both references are required for a pre-stressed buckling analysis.

Stress and Strain Recovery Coefficients Available for CBUSH

The RCV continuation line to the PBUSH data has been added to specify stress and strain recovery coefficients for the CBUSH. These coefficients are multiplied by the element force to create a stress or strain value.

ASCII Results are Now Available for Stress and Strain Results in NLGEOM, IMPDYN and EXPDYN Analysis

For NLGEOM, IMPDYN, and EXPDYN ASCII results for stress and strain for shells and solids are new writing to the `.sty` files when the STRESS(OPTI) and STRAIN(OPTI) are requested. Note that sets are not supported and results for all elements will be output.

Failure Theories can Now be Specified on PCOMPP and PCOMPG Data for NLGEOM, IMPDYN and EXPDYN Analysis

FT can now be specified on the PCOMPP and PCOMPG data when used in NLGEOM, IMPDYN, and EXPDYN analysis. The failure index for composites are written to the `.h3d` file.

CROD Stress Can Now be Recovered in NLGEOM, IMPDYN and EXPDYN Analysis

CROD stress results can now be calculated in NLGEOM, IMPDYN and EXPDYN analysis.

Fatigue Damage and Life Results are Now Written to the HyperMesh .res Results File

Fatigue Damage and Life results are now included in the HyperMesh .res results file.

MPCFORCES Output Written to the .h3d File for Static Analysis

In addition to the .op2 file, MPCFORCES results for static analysis are now available in the .h3d file.

Unreferenced SET Data Written to the .op2 File

Even if a SET is unreferenced by the input data, the SET data is written to the .op2 file. This allows SET data to be passed to and used by post-analysis programs, such as those that calculate fatigue life.

SET Data Written to the .h3d File

SET data (both referenced and unreferenced) is written to the .h3d file.

Negative Values for LDM and Scale Factors Now Allowed in FATLOAD Data

This allows more flexibility in defining fatigue loading parameters.

Drilling Stiffness of Shells Can be Turned Off for Nonlinear Analysis

The data IDRIL has been added to shell element property parameter data (XSHLPRM) to run off the drilling stiffness of shell elements for nonlinear analysis types NLGEOM, IMPDYN, and EXPDYN.

Composite Laminate Added to Account for Offset Elements

The PCOMP/PCOMPG/STACK now support offset elements in laminate option (LAM) for smeared laminates. If the elements are offset (Z0 not equal to one half of the lay-up thickness), use the new laminate option SMEARZ0.

Composite Failure Index and Strength Ratio Now Output to the .op2 File

The composite failure index and strength ration is now output to the .op2 file in the OEFIT and OESRT data blocks.

Geometric Nonlinear Solutions

The following enhancements have been made for the geometric nonlinear solutions, i.e. the NLGEOM, IMPDYN and EXPDYN analysis types.

- It is no longer a constraint that continuing geometric nonlinear subcases must have the same boundary conditions (same SPC reference).
- 2D orthotropic materials are now supported (MAT8).
- Output of internal forces is now supported (MPCFORCE).
- Interdependent coordinate systems are now supported (coordinate systems that reference other coordinate systems via the RID field).
- Added time history output for rods, bars and beams (XHIST).
- Added output time step control to XHIST Bulk Data entry, allowing output time step control for individual XHIST requests. (XHIST).

- Added option to enable small displacement assumption (NLPARMX, TSTEPNX).
- Added option to enable spring back analysis (NLPARMX).
- Added support of fixed time points that the automatic time step control will adhere to. (NLPARMX, TSTEPNX).
- Expanded material library to include LAW0 (MATX0), LAW21 (MATX21), LAW25 (MATX25), LAW28 (MATX28), LAW43 (MATX43), LAW60 (MATX60) and LAW68 (MATX68).
- Added support of contact definitions for 10-noded tetra elements.
- 2D laminated composite definitions are now supported (PCOMP, PCOMPG).
- Orthotropic nonlinear solid properties are now supported (PSLDX6).
- Added support of enforced displacements (SPCD) for NLGEOM analysis type.
- Added support of Rotational Velocity loading (RFORCE data item A).
- Adjusted the default convergence criteria defined on the TSTEPNL card from PW to UPW.
- Added support for rotational displacement output requests (DISP(ROTA)).
- Added support for grid point stress (GPSTRESS) output.
- Added support for cylindrical coordinate systems (CORD2C).
- CBAR and BEAM Von Mises stress is written to the `.h3d` file
- The Force-Deflection Curve from Snap Through analysis can be generated using PARAM,SNAPTHRU,YES

Default Projections and Checks for CWELD Elements

Several adjustments were made to the default behavior of the CWELD element to enhance its ease of use.

New Parameters

PARAM,SNAPTHRU: PARAM,SNAPTHRU,YES is used to control the generation of a Force-Deflection curve from Snap Through analysis. The default is NO

New Subcase Information Entries

- **SUBCOM**: Specifies that results are calculated as a linear combination of preceding static SUBCASE results.
- **SUBSEQ**: Specifies the weighing for the linear combination of preceding static SUBCASE results for the SUBCOM.
- **PRETENSION**: Used to select and activate a pre-tensioning bolt load.

New Bulk Data Entries

- **CFAST and PFAST**: The fastener connects two surface patches defined shell elements or properties (PSHELL). The fastener has a diameter and stiffness that can be defined in a local coordinate system.
- **CTRAIX6**: Defines a three or six noded triangular axisymmetric element with a material reference.
- **CTAXI**: Defines a three or six noded triangular axisymmetric element with a property reference.

- **PAXI**: Defines the material property for the CTAXI axisymmetric element.
- **MAT3 and MATT3**: Define constant and temperature dependent orthotropic material properties for axisymmetric elements CTRIAX6 and CTAXI.
- **PLOADX1**: Defines a pressure load on an axisymmetric element.
- **CGASK6, CGASK8, CGASK12, and CGASK16**: Used to define gasket elements with 6, 8, 12, and 16 grids respectively.
- **PGASK**: Used to define gasket element properties and materials.
- **MGASK**: Used to define nonlinear material properties for gasket elements.
- **PCNTX2, PCNTX5, PCNTX7, PCNTX11 and PCNTX20**: Replace PCONTX for defining contact types for geometric nonlinear (NLGEOM) analysis. PCNTX2 is used to define failure for FREEZE contact. PCNTX11 is used to define edge to edge and line to line contact.
- **CONTX11**: Used to define the Edge to Edge or Line to Line Contact Interface Definition with the PCONTX data.
- **LINE**: Used to define a line for Line to Line Contact
- **MATX25**: MATX25 material definition for support of elasto-plastic orthotropic material with TSAI-WU and CRASURVT yield criteria for composite shell materials has been added for NLGEOM, IMPDYN, and EXPDYN nonlinear analyses.
- **MATX28**: MATX28 material definition for support of Honeycomb materials has been added for NLGEOM, IMPDYN, and EXPDYN nonlinear analyses.
- **PSLDX6**: PSLDX6 property definition for support of additional orthotropic SOLID has been added for NLGEOM, IMPDYN, and EXPDYN nonlinear analyses.
- **CSEAM**: Defines the seam weld element connection data.
- **PSEAM**: Used to define the seam weld element property data.
- **PRETENS**: Defines the ROD, BAR, or BEAM element used to model the pretensioned bolt.
- **PTADD**: Defines a pretension load as a linear combination of load sets defined via PTFORCE, PTFORC1, PTADJST and PTADJS1 entries.
- **PTADJS1**: Defines adjustment (additional shortening) on a set of pretension sections.
- **PTADJST**: Defines adjustment (additional shortening) on pretension section.
- **PTFORC1**: Defines pre-tensioning force on a set of pretension sections.
- **PTFORCE**: Defines pre-tensioning force on pretension section.
- **MATX0**: Void material property extension for geometric nonlinear solutions.
- **MATX21**: Rock-concrete material property extension for geometric nonlinear solutions.
- **MATX43**: Hill orthotropic material property extension for geometric nonlinear solutions.
- **MATX60**: Elastic plastic piecewise nonlinear material property extension for geometric nonlinear solutions.
- **MATX68**: Honeycomb material property extension for geometric nonlinear solutions.
- **PCOMPX**: Laminated composite property extension for geometric nonlinear solutions.

Enhanced Subcase Information Entries

- **STATSUB**: Added type PRETENS to specify bolt pretension
- **TSTRU**: In addition to using the resulting thermal field from a steady state heat transfer analysis for a thermal load on a structural analysis, the thermal field from the last time step of a transient heat transfer analysis can be used for the thermal load

Enhanced Bulk Data Entries

- **PRETENS**: Data to define the location, direction, and type of pretension for 3D bolts was added
- **PBUSH**: The RCV continuation line has been added to specify stress and strain recovery coefficients for the CBUSH
- **PCONT**: The CLEARANCE item in the PCONT data can be used to specify an initial clearance between the master and slave contact surfaces. This does not depend on the actual distance between the surfaces.
- **XSHLPRM**: The data IDRIL has been added in order to turn off the drilling stiffness of shell elements for nonlinear analysis types NLGEOM, IMPDYN, and EXPDYN. The default value is 1 (ON) for NLGEOM and IMPDYN SUBCASES.
- **PCOMP/PCOMPG/STACK**: There is a new laminate option (LAM) for smeared laminates. If the elements are offset (Z0 not equal to one half of the lay-up thickness), use the new laminate option SMEARZ0.
- **MATS1**: The MATS1 data now allows specification of Kinematic and/or Mixed Hardening rules through expanded options for the HR data.
- **DRAPE**: Integer values for T and THETA are now read in correctly as real values.
- **NLPARMX**: Added options to control small displacement assumption, spring back analysis and fixed time points.
- **PBUSHT**: The TYPE KN has been added to define the nonlinear force vs. deflection curve for nonlinear springs.
- **CBUSH1D/PBUSH1D**: Support of the nonlinear 1D bushing element is now available for nonlinear static (NLSTAT) analysis. In previous version, the CBUSH1D element was treated as a linear element during NLSTAT analysis.
- **XSTEP**: The maximum number of iterations (NITER) and output frequency (NPAMS) for the conjugate gradient have been added.

Resolved Issues

- The in-plane shear stress for the CQUAD4 and CTRIA3 was not accurate. In the pure in-plane bending state there was some error in the values for the shear stress.
- In some cases, corner stresses for shell elements could be incorrect when there were also solid elements in the model and corner stresses were requested for those elements, as well.
- In some cases, corner stresses for solid elements could be incorrect.
- Imposed displacement (SPCD) in a local coordinate system was implemented incorrectly for nonlinear analysis types NLGEOM, IMPDYN, and EXPDYN.
- The calculated contact pressure was incorrect for certain meshes.
- GPSTRESS now available for multiple SUBCASE in the `.res` file. In the past, only the GPSTRESS for the first SUBCASE was available in the `.res` file. Now, the GPSTRESS for every SUBCASE is available.
- The DRAPE data items T and THETA should be real numbers. In the previous release, if integer numbers were used they were not correctly converted to real numbers.
- If the Solution Control data EXCLUDE referenced PCOMPP properties, these were not actually excluded from the buckling analysis. Now, PCOMPP properties referenced by EXCLUDE data are excluded from the buckling analysis.
- Since the corner stress is the same as the centriodal stress for CTRIA3 elements, output of corner stress to the `.op2` file was skipped if the model only contained CTRIA3 elements. Now, corner stress

results will be written to the .op2 file, even if the model only contains CTRIA3 elements.

- PS Field on GRID Bulk Data Entry Was Ignored for Geometric Nonlinear Solutions: PS field is now recognized for the geometric nonlinear solutions, i.e. the NLGEOM, IMPDYN and EXPDYN analysis types.
- Loads with Magnitude 0.0 Were Incorrectly Interpreted in Geometric Nonlinear Solutions: Forces, moments, pressures with magnitude 0.0 are now interpreted correctly for the geometric nonlinear solutions, i.e. the NLGEOM, IMPDYN and EXPDYN analysis types.
- Wrong Thickness Used for Contact Involving Laminated Composite Materials: The correct thickness is now being used for contacts involving laminated composite materials.

Back to top

RADIOSS for Noise and Vibrations (Bulk Data Format)

AMSES Large Scale Eigenvalue Solver

The AMSES (Automated Multilevel Substructuring Eigenvalue Solver) can calculate thousands of modes for models with millions of degrees of freedom in less than one hour. The solver scales well with multiple CPU's and can be used for modal frequency response and transient analysis, as well as Noise and Vibration analysis. In addition, it can be used to generate structural and combined fluid-structure CMS Super Elements. AMSES handles unconnected structures, can be used for modal analysis with SPCD enforced motion, and runs on both Linux and Windows operating systems. AMSES is an integral part of RADIOSS and requires no special executable or additional licensing.

Fast Parallel Solver for Modal Frequency Response

A new parallel algorithm has been developed to quickly solve the modal frequency response equations when thousands of modes are used to define the modal space. With nearly linear parallel speed up the solver can handle problems with any combination of modal damping, viscous damping, low and medium rank structural damping in both the fluid and structure. In addition, SPCD enforced motion can be used and modal energy calculation are available. The use of this new method is controlled by the PARAM FASTFR.

Modeling the Effect of Heavy Incompressible Fluids on Structural Vibrations

The MFLUID data can be used to model the effects of heavy fluids like oil or gasoline on the structural vibrations of their containers. This simplified formulation of fluid structure interaction offers an efficient approach to take fluid effects into consideration.

Equivalent Radiated Power and Far Field Sound Level Pressure

The Equivalent Radiated Power from structural panels can now be calculated. The response can be calculated for a set of grids or by panels. In addition, the Intensity can be calculated on the panel and the far field sound pressure level at microphone locations can be output across the frequency range.

One Step Transfer Path Analysis

The Point Mobility, Grid Point Forces, Path Participation, and Transfer functions for multiple control volumes can all be calculated automatically in a single analysis run. With this new algorithm, the solution time is significantly less than either of the solution times in the standard two step process.

MFLUID is Available for Modeling a Virtual Fluid

A fluid bounded by shell elements can be modeled by just specifying the shell elements and fluid properties. The effect of the fluid is taken into account during frequency response analysis.

Automatic One Step Transfer Path Analysis

Using the new PFPATH data to specify the connections, the transfer function, grid point force, and contribution from each connection degree of freedom are automatically calculated during a single run.

Added PARAM,AMSESLM So the Large Mass Method Can be Used with EIGRA

SPCD should be used for enforced motion for frequency response. If the old Large Mass Method is used for enforced motion for modal frequency response and EIGRA is used to calculate the modes, then PARAM,AMSESLM,1 should be specified.

Added PARAM,GE_MOD to Override All Material, Element, and Property GE Data

The value of GE_MOD will replace the value of GE on all MATx, CELAS2, PELAS, PELAST, PBUSH, PBUSHT, PCOMP, PCOMPP, and PCOMPG data. This includes blank values of GE. GE_MOD can be set to zero to remove all structural damping from the model.

Equivalent Radiated Power (ERP)

The Equivalent Radiated Power (Square of the normal velocity integrated over the area) from a panel is calculated and output to the `.pch` file.

Component Dynamic Synthesis (CDS) Super Elements

CDS Super Elements contain the loading frequency information and can greatly reduce the analysis type in residual runs. The loading frequencies in the residual run must be a subset of the frequencies specified during the creation run.

Output of Applied Loads for Frequency Response and Transient Analysis

Applied load output (OLOAD) is now available for Frequency Response, Coupled Fluid-Structure (NVH) Frequency Response, and Transient Analysis.

The OLOAD information can be output to the `.pch`, `.op2`, and `.h3d` results files.

Ability to Control the Output of Results from Interior Points and Elements in Super Elements

The OUTPUT options NODMIG, DMIGALL, and DMIGSET are used to specify that no results, all results, or a SET of results are output for interior points and elements of a CMS Super Element in a residual run. The SET is defined in the corresponding output request data.

Automated Multi-level Sub-structuring Eigenvalue Solver

A state-of-the art automated multi-level sub-structuring method for eigenvalue extraction is now available in RADIOSS.

This feature is available for normal modes, modal frequency response and modal transient response solution sequences, as well as CMS super element generation run.

This solution method is activated when the EIGRA Bulk Data entry is referenced via the METHOD data selector in the Subcase definition. It may also be activated by adding PARAM,AMSES,YES or via the command line argument –amses.

Hybrid Damping for Direct Dynamic Analysis

Hybrid damping is now available in RADIOSS.

This is a means of applying modal damping in a direct frequency response or direct transient response analysis.

This is recommended mainly for residual models, where a large part of the structure is represented by external super elements. This is because of the computational expense of this feature.

It is activated via the HYBDAMP subcase information entry, which in turn references a HYBDAMP Bulk Data entry.

Additionally, the DMIGMOD entry has been expanded to allow hybrid damping to be defined for external super elements only (HYBDAMP will apply damping to both residual structure and external super elements).

Frequency Dependent Spring Property

A frequency dependent spring property is now available in RADIOSS.

Similar to the existing PBUSHT frequency dependent bushing property definition, a PELAST frequency dependent spring property definition has been added. It may be referenced by spring elements (CELAS1, CELAS3).

Speed of the Modal Frequency Response Analysis Solution

The speed of the modal frequency response analysis solution has been dramatically increased for problems that have a large number of modes and structural damping and/or viscous damping.

Results for a SET of GRID Can be Requested from a CMS Super Element

During a residual run, the results from a SET of ASET and interior GRID can be requested. In the past only NODMIG or DMIGALL were allowed on the OUTPUT command. Now DMIGSET is supported.

The Fluid-Structure Coupling Matrix Can Now be Written to the .pch File.

The fluid-structure coupling (Area) matrix can now be output to the `.pch` file in DMGI format using the PARAM,AGGPCH,YES command.

SPCD Data Created for CMS Super Elements for Each Mode During Eigenvalue Analysis

A `.spcd` file is created during an eigenvalue analysis of a model that contains CMS Super Elements. This file contains SPCD data for each ASET GRID and SPOINT for each mode. This SPCD data can be used in a following run to recover the internal results for a specific mode.

PFMODE and PFPANEL Can Write Results to the .h3d, .pch, or Both Files

The results from PFMODE and PFPANEL can be written out to the `.h3d` file, the `.pch` file, or both files.

PFMODE Output Can Now be Determined by Displacement, Velocity, or Acceleration

The option **RTYPE** has been added to the PFMODE data in order to specify DISP, VELO, or ACCE to determine the output of the modal participation results.

RMS Results Only Output Option Added

Previously the only way to get RMS results was to request PSDF output. This created huge output files. Now it is possible to request RMS output only using the output option RMS on the DISP, VELO, ACCE, STRESS, STRAIN, and FORCE data.

Support of Additional Damping in the Interface to FastFRS

The interface to FastFRS now supports GE specified on the material data, as well as both structural and fluid modal damping.

Support of Acoustic Loading in the Interface to FastFRS

The interface to FastFRS now supports ACSRCE loading on acoustic grid points.

PSDF Output to .h3d File Expanded

PSDF output is available to the `.h3d` file for displacement, velocity, and acceleration, as well as solid and shell stress and strain and CBUSH force.

PSDF Output to .op2 File Expanded

PSDF output is available to the `.op2` file for solid and shell stresses.

PSDF Plotting Output Expanded

PSDF output is available to the XYPLOT/XYPUNCH/XYPEAK for displacement, velocity, and acceleration, as well as solid and shell stress and strain and CBUSH, CELAS, CDAMP, and CVISC forces.

CMDE Output Now Accounts for Both Viscous and Structural Damping

In previous versions the CMDE output only accounted for viscous damping. Now, output is also generated for structural damping specified by the GE field on the MATx data.

Specifications of the Matrices Used from H3D DMIG File

By default, all matrices in the H3D DMIG file are used in the analysis. To only use a subset of these matrices, use the K2GG, B2GG, M2GG, and K42GG data to specify which matrices are desired. Any unreferenced matrices will not be used in the analysis if any one of K2GG, B2GG, M2GG, and K42GG are used.

Improved Control of ERP Calculations

PARAM data ERPC, ERPREFDB, ERPRHO, ERPRLF, and RHOCP are added for use in ERP and ERP dB calculations. The value of ERP can now be calculated as a sound level using the speed of sound (ERPC), the air density (ERPRHO), and the radiation loss factor (ERPRLF). The value can be converted into decibels using the scale factor (RHOCP) and the reference pressure value (ERPREFDB).

ACMODL Statistics Written to the .out File

The maximum, average, and mean number of structural grids associated with any fluid element face is written to the `.out` file.

EIGRA Solution Time Reduced for Large Eigenvalue Problems

The solution time has been reduced when EIGRA is used for calculation of a large number of modes.

EIGRA Can be Used to Solve Larger Problems

The number of modes that can be calculated when using EIGRA has been increased for large models.

EIGRA and AMLS Can Now be Used for Pre-Stressed Normal Modes Analysis

STATSUB(PRELOAD) can now be used in a SUBCASE that references EIGRA data, or if PARAM,AMLS,YES is present.

Modal Frequency Response Solution Time Reduced for Problems Without Structural or Viscous Damping

If there is only modal damping (no structural or viscous damping), then the resulting modal Frequency Response problem is diagonal. In this case, because a diagonal solution algorithm is used, the solution time is reduced significantly.

AVL/EXCITE Interface

The AVL/EXCITE interface has been enhanced to allow users to obtain results for interior points in a residual model run.

After running an AVL/EXCITE analysis an `.INP4` file is generated that contains the modal participation factors. The `.INP4` file can be used along with the original H3DDMIG file to recover stresses, strains, displacements, velocities, and accelerations from a residual run.

SIMPACK Interface

The SIMPACK interface has been enhanced to allow the generation of flexible bodies using CSET data with the CBN CMSMETH method.

Allow Craig-Chang Super Elements in Residual Run Using AMLS/AMSES

It is now possible to use super elements generated using the General Method (GM) with free boundary interfaces (the Craig-Chang approach), in a residual run with AMLS/AMSES.

Time Varying W3 and W4 for Transient Response Analysis

The TSTEP card has been enhanced to include fields for W3 and W4, such that these quantities can be defined differently for each set of time increments.

Static Loads Can Now be Applied Dynamically for Transient Analysis

The TLOAD1 and TLOAD2 EXCITED data can now reference static loads such as FORCE, FORCE1, MOMENT, MOMENT1, RFORCE, GRAV, and PLOADi.

Modal Participation Ratio Output

Modal participation ratios are output to the `.out` file when PARAM,EFFMASS,YES is defined.

AUTOSPC for AMLS Solver Interface

PARAM,AUTOSPC is now used to control AUTOSPC functionality for AMLS solver interface also.

The parameter previously used to control this, PARAM,AMLSAPC, is now defunct.

SET Data Specification for Rigid Element Output on the MODEL Data

The MODEL data that is used to specify what is stored in the CMS super element/flex body now allows for SETs of rigid elements. Before this enhancement, only RIGID or NORIGID was allowed.

Modal Stress Data Can be Exported Using PARAM,LMSOUT

Setting **PARAM**,**LMSOUT** to **STRESS** will turn on the output of the modal stresses in addition the condensed modes and full diagonal mass matrix.

Element FORCE Results are Available for CBUSH, CELASi, CDAMP, and CVISC Elements in Transient Response Analysis

FORCE results due to damping are calculated for the CBUSH, CELASi, CDAMP, and CVISC elements during Transient Response Analysis and are written to the .h3d, .op2 and .pch files.

Element FORCE Results are Available for CBUSH, CELASi, CDAMP, and CVISC Elements in Frequency Response Analysis

FORCE results due to damping are calculated for the CBUSH, CELASi, CDAMP, and CVISC elements during Frequency Response Analysis and are written to the .h3d, .op2 and .pch files.

Element FORCE Results are Output from Eigenvalue Analysis

The modal FORCE results in eigenvalue analysis can now be output to the .h3d, .op2, and .pch files.

Element Strain Energy (ESE) is Now Written to the .op2 File

ESE output is now available in the .op2 file. For modal analysis, the ESE output is output as a static subcase for each mode.

Element Strain Energy (ESE) for Frequency Response is Now Written to the .pch File

ESE output is now available in the .pch file for frequency response analysis.

Element Energies (ESE/EKE/EDE) for DMIG (CMS SE) Elements are Now Written to the .out File

The element energies associated with super elements are now output to the .out file.

MPCFORCE Output is Now Available from Modal Frequency Response Analysis

MPCFORCE output is now available from Modal Frequency Response Analysis in the .op2, .pch, and .h3d files.

A File to Allow Optimization of CMS Super Elements Can be Generated

Using the DMIGDV continuation of the CMSMETH data, the contents of an include file used for optimization of the Super Element modal frequencies and their corresponding structural viscous damping values is specified.

Sets of RIGID Elements Can Now be Specified with the Bulk Data SET Data

The set TYPE can now be specified as RIGID in the Bulk Data SET data. This is required when specifying the set of RIGID elements attached to the connection points in the PFPATH data.

New Parameters

- **PARAM,FASTFR**: PARAM,FASTFR is used to control the solution method for modal frequency response analysis. The default is AUTO, but YES and NO can be used to turn on or off the new fast solution method.
- **PARAM,AGGPCH**: Setting AGGPCH to YES causes the fluid-structure coupling (Area) matrix to be output to the .pch file in DMIG format.
- **PARAM,ERPC**: Used to specify the speed of sound in the fluid. This is required to convert the ERP value to a sound level.
- **PARAM,ERPRLF**: Used to specify the radiation loss factor. This is required to convert the ERP value to a sound level.
- **PARAM,ERPRHO**: Used to specify the fluid density. This is required to convert the ERP value to a sound level.
- **PARAM,ERPREFDB**: Used to specify the reference pressure. This is required to convert the ERP value into decibels.
- **PARAM,RHOCP**: Used as a scale factor in the calculation of the ERP value in decibels.
- **PARAM,GE_MOD**: used to specify the structural damping GE value for all MATx and PCOMPx data. This new value will override the value on the MATx and PCOMPx data. This new value can be set to zero. In addition, GE_MOD can be set to NO_GE. In this case no structural damping is considered from MATx, PCOMPx, CELAS, PELAS, and PBUSH data. PARAM,G can be used to specify structural damping based on the entire stiffness matrix.
- **PARAM,NUMEG**: used to specify the estimated number of modes for modal Frequency Response and Transient Analysis, when the number of modes is not specified on the EIGRL/EIGRA data. This number of modes is used to estimate the disk space usage.
- **PARAM,AMSESLM**: SPCD should be used for enforced motion for frequency response. If the old Large Mass Method is used for enforced motion for modal frequency response and EIGRA is used to calculate the modes, then PARAM,AMSESLM,1 should be specified.
- **PARAM,INTRFACE**: This parameter is used to turn off the generation of the .interface file. The .interface file contains the data about the fluid-structure coupling and is created by default. PARAM,INTRFACE,NO will turn off the generation of this file.
- **PARAM,ENFMOTN**: This parameter is used to control the displacement output during modal frequency response with enforced motion. If set to ABS, the absolute displacement is output, and if set to REL, the relative displacement to the enforced motion is output. The default is ABS.
- **PARAM,AMSES**: Switches eigenvalue extraction method from Lanczos to AMSES.
- **PARAM,VMOPT**: This parameter is used to control the method used when the mass from virtual fluid (MFLUID) is used to model incompressible fluids.
- **PARAM,ASCOUP**: Setting this parameter to NO will turn off the generation of the fluid-structure coupling (Area) matrix. This is required when this matrix is read in from the PUNCH file.
- **PARAM,INTRFACE**: Setting this parameter to NO will turn off the generation of the fluid-structure coupling .interface file which is used to check the fluid-structure coupling as determined by the ACMODL data.
- **PARAM,LMSOUT**: Setting this parameter to STRESS will turn on the output of the modal stresses in addition the condensed modes and full diagonal mass matrix.

New Subcase Information Entries

- **MFLUID**: Used to specify the MFLUID Bulk Data SID
- **HYBDAMP**: Hybrid damping data selector. This data selector references a HYBDAMP Bulk Data entry.

Enhanced Subcase Information Entries

- **MODEL**: In addition to RIGID and NORIGID, a SET of rigid elements can be specified to be included in the CMS super element/flex body for display in the residual run.

New I/O Option Entries

- **PFPATH**: References the PFPATH Bulk Data to specify output for One Step Transfer Path Analysis
- **ERP**: Used to request ERP results to the `.pch` file.

New Bulk Data Entries

- **PFPATH**: Used to specify the connections for output of the transfer function, grid point force, and contribution from each connection degree of freedom for automatically One Step Transfer Path Analysis.
- **MFLUID**: Defines the boundaries and parameters of the fluid volume.
- **BNDFIX1, BNFRE1, and BSET1**: Alternate forms of BNDFIX, BNDFREE and BSET data are now available to specify a list of grids all using the same DOF for CMS Super Element creation.
- **ERPPNL**: Used to define the panels for the ERP calculation.
- **PANELG**: Used to define panels for the PFPANEL, PFMODE, and ERP calculations.
- **EIGRA**: Requests real eigenvalue extraction using the Automated Multi-level Substructuring Eigenvalue Solver (AMSES).
- **HYBDAMP**: Defines the application of modal damping in a direct frequency response or direct transient response analysis.
- **DMIGMOD**: Defines the application of modal damping in a direct frequency response or direct transient response analysis to a CMS Super Element.
- **DMIGMOD**: Used to replicate a CMS Super Element in a new location. This allows for the multiple re-use of a single CMS Super Element.
- **CDSMETH**: Used to generate Component Dynamic Synthesis (CDS) Super Elements. These Super Elements contain the loading frequency information and can be much faster than CMS Super Elements in the residual run.

Enhanced I/O Option Entries

- **PFMODE and PFPANEL**: The OUTPUT option can be set to H3D, PUNCH, or BOTH.
- **DISP, VELO, ACCE, STRESS, STRAIN, and FORCE**: The option RMS has been added to request RMS results only.
- **ASSIGN**: Added EXCINP option to reference modal participation factors from AVL/EXCITE, in order to recover results for interior grids and elements.

Enhanced Solution Control Entries

- **RESVEC(NOAPPL)**: The NOAPPL option turns off residual vector generation due to loads. This turns off both the UNITLOD and APPLOD residual vector generation.

Enhanced Bulk Data Entries

- **CMSMETH**: The ability to specify the solver to use (LAN for Lanczos and AMSES for AMSES) has been added. In addition, the Lanczos SHFSCL and AMSES AMPFFACT parameters have been added.
- **CMSMETH**: Using the DMIGDV continuation of the CMSMETH data, the contents of an include file used for optimization of the Super Element modal frequencies and their corresponding structural viscous damping values is specified
- **DRESP1**: Added FRERP for defining equivalent radiated power as a response.
- **PANELG**: Generic panel definition for ERP or panel participation output.
- **ERPPLN**: Panel definition for equivalent radiated power output.
- **TSTEP**: Added W3 and W4 fields allowing these values to be defined differently for each set of time increments.
- **TSTEPNX**: Added options to control small displacement assumption and fixed time points.
- **SET**: Added the TYPE RIGID to define sets of RIGID elements. These sets can be used on the PFPATH data.

Resolved Issues

- The MPC and rigid element force contribution in GPFORCE output for frequency response could be incorrect. Now correct contributions from MPC's and rigid elements are output in the GPFORCE results for frequency response.
- The MPCFORCE output for frequency response could be incorrect.
- Temperature dependent Structural Damping is now available. The T(GE) data on the MATT1, MATT2, MATT3, MATT8, and MATT9 is now supported.
- Use of PARAM,LFREQ in modal transient analysis produced incorrect results. Now correct results are obtained in modal transient analysis when PARAM,LFREQ is used.
- Modal transient analysis with enforced motion (SPCD) gave poor results. A new algorithm was implemented for modal transient analysis with enforced motion (SPCD). Now the results match those from direct transient analysis.
- Unless DISP(ROT)=ALL was requested, the GPKE values were wrong when AMLS or EIGRA was used. This has been corrected.

Back to top

RADIOSS for Kinematic and Dynamics (Bulk Data Format)

Ply-based Composite Definition Can Now be Used in MBD Analysis

PCOMPP ply based composite property data can now be used in the Multi-body Dynamic analysis using rigid bodies (PRBODY) and flexible bodies (PFBODY).

CB and CC Flexbody Generation Time Reduced

The CB generation time is reduced for problems with large number of ASET DOF. The CC generation time is greatly reduced, if AMSES or AMLS is used.

Pre-loaded CB and CC Flexbodies Can be Generated

A LOAD LID and SPC SID can be specified on the PRELOAD continuation of the CMSMETH data to preload the flexbody. The effect of the pre-load on the flexbody is taken into account when the flexbody matrices are calculated.

Residual Vectors Can be Used to Increase the Accuracy of CB and CC Flexbodies

A LOAD LID can be specified on the LOADSET continuation of the CMSMETH data to generate residual vectors that can increase the accuracy of the flexbody results. The degrees of freedom of the loads are used to create residual vectors.

CB and CC Flexbodies Can Contain Reduced Loads

Loads acting on the connection points of a CC or CB flexbody can be generated by specifying a LOAD LID LOADSET continuation of the CMSMETH data. The loads are reduced from the full mesh to the connection points and can be used in the multibody analysis.

CMS Flexbodies Can Now be Created Based on PUNCH DMIG Input Data

The use of PUNCH DMIG input data is now allowed in the input data for CMS Flexbody creation.

1D Element Forces Included in Flexbody Output

The FORCE output request may be used in a flexbody generation run to obtain 1D element force results in the `flexh3d` file.

Enhanced Bulk Data Entries

- **CMSMETH**: A LOAD LID and SPC SID can be specified on the PRELOAD continuation to preload the flexbody. The effect of the pre-load on the flexbody is taken into account when the flexbody matrices are calculated.
- **CMSMETH**: A LOAD LID can be specified on the LOADSET continuation to generate residual vectors that can increase the accuracy of the flexbody results. The degrees of freedom of the loads are used to create residual vectors.

Back to top

RADIOSS - General (Bulk Data Format)

GRID Data Replication

GRID data can be generated using replication markers (=, ==, and *). This allows for the generation of GRID data where the ID, coordinate systems, and coordinates are incremented for each new GRID.

AUTOSPC DOF Written as SPC/SPC1 to a PUNCH File

The AUTOSPC(PUNCH) command will initiate the generation of a _spc.pch file that contains SPC/SPC1 data for the AUTOSPC DOF for each SUBCASE.

The SYSSETTING Command SAVEFILE Has Been Added to Save Analysis Results from Previous Runs

The new command SYSSETTING(SAVEFILE=ALL) will save the previous run's results files (.h3d, .op2, .pch, etc.) by renaming the old file with a numeric extension as is currently done with the .stat and .out files. This can also be set in the configuration file (hwsolver.cfg).

-compress Option Now Works with a Tolerance

A tolerance can be specified with the –compress option to combine property data with the same or similar values.

Solution Control AUTOSPC Command

The new Solution Control AUTOSPC command replaces the AUTOSPC and PRGPST PARAM data.

GENEL Element

The General Element (GENEL) for which the stiffness matrix for an arbitrary number of grids has been added. Either the stiffness or flexibility matrix can be input by the user.

No Available License, No Worries

If a RADIOSS run is submitted with -licwait, but there are not enough HyperWorks Units available for the job to start, the job will wait until enough HyperWorks Units are available, and then it will check them out and start running.

Bulk Data SET1 and SET3 Formats Supported

SET1 and SET3 set definitions are now supported for defining sets in the Bulk Data section. These are internally converted to the more powerful and flexible Bulk Data SET data.

Passing of Metadata from the Input File to a Resulting XML File

If the keywords METADATA and ENDMETADATA are present in the input file, the content between the keywords is written in a separate <basename>_metadata.xml file. This allows passing of metadata information seamlessly through the solver.

New 3D Orthotropic Material Definition (MAT9ORT)

A solid orthotropic material definition (MAT9ORT) using engineering properties rather than stiffness matrix input is now available in RADIOSS.

This is an alternative input format for the MAT9 anisotropic material definition and is translated on reading. The echoed material definition will use the MAT9 definition format.

Automatic Input Update

An automated mechanism for updating input definitions from an external file is now available in RADIOSS.

This allows optimization output from OptiStruct (e.g. .grid file, .prop file) to be used in a follow-on run without the need to regenerate the input file.

The I/O Option ASSIGN has a new sub-option UPDATE, which allows an external file to be referenced for the purpose of updating input. Input definitions in the referenced file are then used in place of the definitions in the source file.

Another I/O Option, UPDATE, has several parameters to control the behavior of ASSIGN,UPDATE.

Long-field Free Format Input

RADIOSS now accepts Bulk Data input using the long-field free format.

Long-field free format consists of four comma separated input fields per line. This input format is identified by an asterisk in the first column or (for the line with the card name) at the end of the card name.

Compressed Input Files

RADIOSS now accepts compressed input (using GZIP format) directly.

The main input file and/or any include files may be compressed using the GZIP format. Compressed files are recognized by the .gz extension. It is not necessary to modify INCLUDE lines in the input deck with this .gz extension.

Note: Files compressed using Windows ZIP format are not recognized.

Contributions from Individual MPC and Rigid Elements are Listed in the GPFORCE Output

In the GPFORCE balance output to the .op2 file the contributions from individual MPC and rigid elements are listed separately. This allows the HyperMesh Free Body Diagram (FBD) tool to correctly list the individual contributions. They are also listed individually in the .pch file output.

DMIG Mass Effect is Included in the Generation of RFORCE and GRAV Loads

The mass matrices in the H3D DMIG and PUNCH DMIG files are used when calculating RFORCE and GRAV acceleration loads. The mass contribution to these loads can be turned off with the PARAM,CMSALOD,NO data.

WARNING Messages Can be Turned Off

By setting the limit to NONE in the MSGLMT data the WARNING will no longer be issued.

Shell Element Offset can be Specified as TOP or BOTTOM

In addition to the actual offset distance, the ZOFFS data can be set to keywords TOP and BOTTOM to offset the plate elements by ½ of their thickness in the appropriate direction.

Stress and Strain Results for CBAR, CBEAM, and CROD are Now Written to the .pch and op2 Files

Stress and Strain results for CBAR, CBEAM, and CROD are now written to the .pch and .op2 files for static, frequency response, and transient analysis.

SPCFORCE Output is Now Available for NLGEOM, IMPDYN and EXPDYN Analysis

SPCFORCE output is now available for NLGEOM, IMPDYN and EXPDYN analysis. These results are written to the .h3d file.

SPCFORCE Output is Now Available for Eigenvalue Analysis

The SPCFORCE for each mode shape can now be output to the .pch and .h3d files.

MAT9ORT Can Now be Used in NLGEOM, IMPDYN and EXPDYN Analysis

MAT9ORT referenced by PSOLID can now be used in NLGEOM, IMPDYN and EXPDYN analysis.

PARAM,ALPHA1 and PARAM,APHA2 Can Now be Used in NLGEOM, IMPDYN and EXPDYN Analysis

Rayleigh damping can now be used in NLGEOM, IMPDYN and EXPDYN analysis.

Multiple XDAMP Data Can be Used in EXPDYN Analysis

Multiple XDAMP data can be used for define Rayleigh damping in EXPDYN analysis.

Shell Elements have Drilling Degree of Freedom Stiffness in NLGEOM and IMPDYN Analyses

Shell elements now have drilling degree of freedom stiffness in NLGEOM and IMPDYN analyses.

The Shell Element Thickness is Now Stored with the Model Information in the .h3d File

The shell element thickness is stored with the model information in the .h3d file.

Structural Damping and a Stress Recovery Coefficient Can Now be Specified on CELAS4 Data

GE and S can now be specified on CELAS4 data to be consistent with CELAS2 data.

AMSES Statistics Written to the .stat File

For the AMSES solver, the number of analysis DOF, output DOF, residual vectors, and auxiliary vectors are written to the .stat file.

Mass Properties Written H3D CMS Super Element Files

Mass properties are written to the H3D CMS Super Element files. This allows for printing of the system mass that includes the CMS Super Element mass in the .out file for residual runs.

Rigid Elements are Now Included in the .mnf File

Rigid elements (RBE1, RBE2, RBE3, RROD, and RBAR) are now included in the model information in the .mnf file.

Support of Problems with Many Constraint Equations Due to Large RBE3 Elements

Analysis of models with a large number of large RBE3 elements requires the reductions of many multi-point constraint equations. Now this reduction step can handle even a larger number of constraint equations.

Scratch Files Created by FastFRS are Now Deleted at the End of the Run

In the past, the FastFRS scratch files remained in the directory and had to be removed by the user. Now, the scratch files created by FastFRS are deleted at the end of the run.

Disk Space Estimate Improved for Modal Analysis

Previously, the disk space estimate for modal Frequency Response and Transient Analysis was based on just 20 modes, if the number of modes was not specified on the EIGRL/EIGRA data. Now the default is 1000 modes. In addition, PARAM,NUMEG can be used to specify the estimated number of modes. This estimated number of modes is used to determine the disk space usage.

Support of Platform MPI Version 8 on Windows

RADIOSS will now run MPI parallel on Windows machines that have Platform MPI version 8.

Element Types, Properties, and Materials Can Now be Used to Define Sets of Grids Using Bulk Data SET

The Bulk Data SET definition can include element types (PLOTEL, CBEAM, etc.), property types, and materials to define a set of GRID points associated with those elements, properties, or materials.

RIGID Option for Stiffness Entry on Bushing Property Definition

A new keyword option has been added for stiffness entries on PBUSH property definition.

Instead of entering a real value for any of the stiffness components, the user may now enter the keyword RIGID.

When RIGID is entered, a very high relative stiffness value (quasi-rigid) is used for that stiffness component.

Improved SMP Parallelization for Iterative PCG Solver

Improvements were made to the performance of shared memory parallelization for the iterative PCG solver.

Solver Run Manager GUI

Several enhancements have been made to the HyperWorks Solver Run Manager GUI.

- Checkboxes were added to facilitate SMP and SPMD parallel runs.
- GUI will remember options used for Bulk Data input and Block input separately, and based on the selected input file type the appropriate previously used options will be selected and displayed.
- Added a “Results” button that appears once the job has completed. This button will launch HyperView and load the resulting `.mvw` file. There are also additional options under the View menu, to view the resulting `.html` summary file and the resulting `.h3d` results file.

PBEAML Can be Used with HyperBeam

In addition to PBARL, PBEAML can now be used with HyperBeam.

Duplicate Property and Material Data is Allowed

Duplicated property and material data is now allowed in the input data.

New Parameters

- **PARAM,CMSALOD**: PARAM,CMSALOD is used control the use of the H3DDMIG and PUNCH DMIG mass matrices when generating RFORCE and GRAV loads. The default is YES.
- **PARAM,RENUMOK**: PARAM,RENUMOK is used turn on automatic renumbering of inside out solid elements. It is preferable to load the model into HyperMesh and then export it to get the grid number sequence to be correct.

New SYSSETTINGS Entries

- **DUPTOL=0,1,2,3,4,5**: In addition to setting with PARAM data, DUPTOL can now be used with SYSSETTING to specify the number of significant digits to ignore when checking for duplicate GRID and CORDx data.
- **SAVEFILE=ALL**: The new command SYSSETTING(SAVEFILE=ALL) will save the previous run's results files (`.h3d`, `.op2`, `.pch`, etc.) by renaming the old file with a numeric extension, which is currently done with the `.stat` and `.out` files.

New I/O Option Entries

- **Update**: Controls the behavior of the input update feature where an external file can be selected to update input definitions in the current file (ASSIGN,UPDATE).

New Solution Control Entries

- **AUTOSPC**: Replaces the AUTOSPC and PRGPST PARAM data.

New Subcase Information Entries

- **P2GSUB**: Used to specify a specific loading vector to be pulled from DMIG data and applied as a load in a single SUBCASE. For example, to apply the second load vector of DMIG PAX in a specific SUBCASE, use P2GSUB=PAX(2) data in that SUBCASE.

New Bulk Data Entries

- **GENEL**: Used to define a general finite element stiffness matrix to be added to the analysis.
- **METADATA / ENDMETADATA**: The content between the keywords METADATA and ENDMETADATA in the input file is written to the `<basename>_metadata.xml` file. This allows passing of metadata information seamlessly through the solver. METADATA/ENDMETADATA pairs can exist in both the Solution Control Section, as well as the Bulk Data section of the input file.
- **MAT9ORT**: A solid anisotropic material definition using engineering properties rather than stiffness matrix input.

Enhanced PARAM Entries

- **DUPTOL**: PARAM,DUPTOL can now use values from 0-5 to specify how many significant digits to ignore when checking for duplicate GRID and CORDx data.

Enhanced I/O Option Entries

- **OUTPUT**: CMS Super Element results output can be controlled by SET data using DMIGSET.
- **MSGLMT**: Setting the limit for any message to NONE is a special case - it will prevent the message

from being printed.

- **MSGLMT**: Setting the limit for any message to 1 is a special case - it will prevent any information about consecutive occurrences to be registered and counted.
- **ASSIGN**: Added UPDATE option to reference updated property input.
- **SYSSETTING:** Added H3DVTAG option.

Enhanced Solution Control Entries

- **AUTOSPC(PUNCH):** The PUNCH option on the AUTOSPC data will initiate the generation of a `_spc`.pch file that contains SPC/SPC1 data for the AUTOSPC DOF for each SUBCASE.
- **SOLVTYP SOLVER AUTO option**: The AUTO option has been added to the SOLVTYP SOLVER data. This option causes RADIOSS to automatically choose between the direct and iterative solver to be used as the solver for linear static and geometric nonlinear analysis.

Enhanced Bulk Data Entries

- **CELAS4**: Structural Damping (GE) and a stress recovery factor (S) can now be specified.
- **PSHELL, CQUAD4, CTRIA3, CQUAD8, and CTRIA6**: In addition to the actual offset distance, the ZOFFS data can be set to keywords TOP and BOTTOM to offset the plate elements by ½ of their thickness in the appropriate direction.
- **DTI,UNITS**: Expanded the set of available units.
- **PBUSH**: Added RIGID keyword option for stiffness.
- **XHIST**: Added DDTHM field, allowing output time step control for individual XHIST requests.
- **XHIST**: Expanded the available types, adding BAR, BEAM and ROD.
- **SOLVTYPE**: Added the MUMPS solver type.

Resolved Issues

- The signs of the MPC Force had been reversed in the .op2 results file.
- The mass on the PBUSH1D was ignored.
- Error Termination When Unused Property References a Non-existing Material: We no longer terminate that solution when an unused property references a non-existing material. Instead we just issue a warning.

Back to top

RADIOSS for Crash and Safety (Block Format)

Tied Contact with Penalty Formulation (Spotflag=25)

Tied contact with penalty formulation resolves incompatible kinematic conditions on slave nodes. For example a slave node might have 2 master surfaces, or belong simultaneously to tied interface and to a rigid body, or any other kinematic condition, like boundaries conditions, imposed displacement, etc.

Tied Contact with Rupture (Spotflag from 20 to 22)

Rupture is enabled inside tied contact (kinematic formulation). Maximum force in the contact can be defined separately in normal and tangential direction with two curves:

1. Tensile stress versus normal elongation
2. Shear stress versus tangential displacement

Rupture is defined accordingly with maximal normal elongation and/or maximal tangential displacement.

Limitations: not compatible with /DT/NODA/CST.

HEPH Improved Hourglass Control

HEPH is an under-integrated element in which the hourglass stiffness is computed for the actual stress state, as opposite to the elastic modulus. This makes it a very efficient and stable element for elasto-plastic materials, but with some limitations for materials like rubber, foams, etc. when we go to the highly nonlinear portion of stress-strain curve.

Augmented hourglass formulation computes hourglass stiffness directly on the stress-strain curve prescribed at the material input level. It is available both for:

- rubber like materials, hyper-elastic and hyper-viscoelastic materials (LAW42, 62, 69, 82)
- viscoelastic foam materials (LAW38)

Joints

There is a new penalty joint with simplified input and augmented robustness. Input is based on new property (/PROP/KJOINT2) and a new spring defined with 2 to 6 nodes. Joints skews definition is very simple, as it is based on spring nodes. Joints stiffness can be defined by user, or computed automatically (default).

Supported types are: cylindrical, revolute, translational, spherical, rigid, free, planar, and Oldham.

Optional features: friction, stop angles, blocking/unblocking thru sensors, automatic stiffness computations in the fixed directions, etc.

Note: Universal Joint will be supported in next patch (12.0.210)

Crushable Beam (/PROP/TYPE44)

2 nodes spring element which crushing behavior is prescribed through functions (loading and unloading). It can replace beam type structures, for quick solutions in parametric/optimization studies.

Springs and Beams

New options are available for spring TYPE4, TYPE8, TYPE12, TYPE13 and TYPE25.

- Damping can be prescribed as a function of the deformation speed.
- New hardening flags with (1) isotropic hardening and nonlinear unloading (2) elastic hysteresis

---

Transformations and Rotations of Submodels

Automatic positioning of submodels has been improved by supporting transformations and rotations for all options, including the ones which input is not based on nodes (like vectors, orthotropic properties, initial/boundary conditions, rigid walls, boxes, etc).

Airbags

Several enhancements to improve results quality and solution robustness:

1. /LEAK/MAT: defines porosity for fabric materials (both LAW19 and LAW58). Several leakage models available, including dependence of pore size on airbag over-pressure.
2. /MONVOL/FVMBAG1: Injector properties can be defined referring to /PROP/INJECT card (as for /AIRBAG1); gas mixtures and properties can be prescribed referring to /MAT/GAS cards.
3. /MONVOL/FVMBAG and /MONVOL/FVMBAG1: FVM mesh can be applied on the reference geometry (flag Iref). This is currently compatible with /REFSTA only (not yet with XREF).
4. Dt min flag (/INTER/TYPE7, 11, 19): in case the time step is fixed by an impacted node, the same is removed from the selected contact interface. It is equivalent to /DT/INTER/DEL defined in the Engine file, but it acts on a single interface only.
5. /LAW58 and /LAW19: sensor activation for reference metrics (flag Isens), in case inflation occurs at time>0.

Back to top

RADIOSS for Metal Forming (Block Format)

AMS Compatibility with Mesh Adaptivity

Adaptive mesh is a common practice in Metal Forming applications. The Advanced Mass Scaling technique in combination with mesh adaptivity, significantly reduces the elapsed time.

Contact TYPE21

- New input: Rigid bodies can be coupled with TYPE21; this allows higher usage flexibility
- Support of thermal radiation.
- Heat conductance depending on contact pressure.
- Limitation of the friction force with reference to shear stress.

Material LAW78

This is the Yoshida-Uemori material law; it provides higher accuracy for spring-back prediction for metal forming applications. It takes into account Bauschinger and cyclic hardening effects.

Initialization of Stress/Strains (Reading & Output) for Brick Elements

/STATE/BRICK/STRAIN/GLOBFULL and /STATE/BRICK/STRESS/GLOBFULL

- Applies initial strain/stress state in global reference system

/INIBRI/STRA_FGLO and /INIBRI/STRS_FGLO

- Initial strain/stress for orthotropic materials (output in global coordinate system)

/STATE/BRICK and PID22

- Applies an initial strain/stress state in the global reference system.

Initialization of Nodal Temperature (Reading & Output)

/STATE/NODE/TEMP

/INI/NODE/TEMP

Back to top

RADIOSS for Blast Simulation (Block Format)

Material LAW5

This is JWL material law. New flag “Eadd” allows to treat after burning energy produced by some explosives.

Material LAW51

Qa, Qb and hourglass coefficient can now be defined at the PID level for the corresponding part. This improvement ensures the results consistency between LAW5 and LAW6.

Material LAW79

This material law describes the behavior of brittle materials such as ceramics and glass. The implementation is the second Johnson-Holmquist model.

/TH/BRIC/BFRAC

Burn fraction output for LAW51.

Back to top

RADIOSS - General (Block Format)

Time Step Control for Contact Interfaces (Advanced Mass Scaling)

Time step control for contact interfaces (Types 7, 10, 11, 19 and 20) switch to AMS. No mass is added during the computation. It’s effective anytime the time step is bounded by the contact interface.

Limitations: not compatible with /DT/NODA.

/INTER/TYPE7

Initial Penetration Treatment - New flag Fpenmax deactivates node stiffness, whatever the value of INACTI flag, if % of penetration into the gap is larger than prescribed value.

/INTER/TYPE7, TYPE11 and TYPE19

Dtmin flag can be activated for each individual contact; contact stiffness on nodes which are limiting time step is deactivated. It behaves same as the /DT/INTER/DEL available at Engine level for all contacts.

/INTER/TYPE7, TYPE10, TYPE11, TYPE19 and TYPE20

Automatic memory resizing; MultiMP user input is no longer required and it is ignored.

Material LAW2 (Johnson-Cook)

New formulation with kinematic hardening based on Ziegler-Prager model, is available. Accuracy is improved specially when material is subjected to cyclic loading-unloading, as Bauschinger effect is taken into account. Mixed mode hardening (full isotropic/kinematic) is also available.

Material LAW25 (Composite)

Max plastic work per ply and per direction.

Solid Elements

Navier Stokes viscosity model is available for all material laws.

Pressure Loads

Simplified pressure effects modeling on a lagrangian structure. It can be applied to emulate static and fluid-dynamic loadings. For example, hydrostatic pressure, air-flow pressure, etc.

Centrifugal Loads

Prescribes acceleration fields for rotating structures such as propeller blades, turbines, etc. Centrifugal loads can be computed using either linear or nonlinear solution (block format). Typical application is centrifugal loads initialization (implicit) of propeller blades before bird strike (explicit).

/INIVEL/AXIS (Starter)

Initialize both translational and rotational velocities on a group of nodes in a given coordinate system; the axis is defined using a frame.

/INIV/AXIS (Engine)

Initialization of both rotational velocity about an axis and translational velocity is enabled in the engine file also; it allows initializing velocities using an existing restart file (for example after gravity settings run).

Cylindrical Coordinate System

It simplifies the input for axi-symmetric imposed displacements, velocities and accelerations, etc. Available within fixed/moving skew and frames.

Input Parameterization

Supported parameters are: integers, reals, text and mathematical expressions, using Templex from HyperWorks.

Model Sensitivity (/DAMP/INTER)

Knowing and managing model sensitivity is of primary interest to ensure accuracy and quality numerical results analyses. One simple example is evaluation of design changes: it is actually hazardous to conclude if a design change is effective or not without knowing the amplitude and position of the sensitivity corridor (generated by some noise). In other words, the design change may have an effect which is lower (or not) than the one produced by just some noise in the model.

In explicit computations, contact has been identified as one of the critical numerical treatments generating results scatter; fine tuning of contact algorithms, and in particular damping the rotational degree of freedom of a node during the time in which it is subjected to contact, allows to manage results scatter induced by numerical noise.

Single Precision

Improved performance for the Extended Single Precision version.

Starter Parallelization (SMP)

Starter SMP parallelization has been achieved. Parallel Starter mode is activated using the same environment variables as Engine or by command line, using `-nthread (-nt)` option.

Engine Parallel Performance (SPMD)

Contact TYPE7 improved scalability; the larger the model, the better the scalability improvement (or solution speed if using same number of cores).

SPH INLET & OUTLET Parallelization (SPMD)

Massive SPH models (10M+ particles) can be resolved. Enables the SPH technique to solve complex transient problems in different application fields, such as Fluid Dynamics, Fluid Structure Interaction, etc. The maximum number of neighboring particles to be computed can be defined; setting it to a lower value than the maximum number to be stored, allows the reduction of the sorting frequency in some extreme situations.

Limitations: /SPH/INLET is not /PARITH/ON compatible yet.

GPU Support for Implicit Iterative Solver (Hybrid Intel MPI)

Supported on Nvidia Tesla under Linux64.

Anim Output

- **/ANIM/SHELL/PHI** and **/ANIM/SHELL/PHI/n** - Output of first orthotropy direction for composite shells.
- **/ANIM/VECT/FREAC** and **/ANIM/VECT/MREAC** - Output of reaction forces and moments in animation files on nodes with boundary conditions, imposed velocities, displacements and accelerations.

/ANIM/NODA/DAMA2

Damage output for interface TYPE2.

Time History Output

/TH/NODE/REAC - Compatible with /IMPVEL, /IMPDISP, and /IMPACC

ABF (Altair Binary Format) Output

Time history data can be generated in ABF native format by ABFconverter program; it might be useful to plot new TH variables, eventually not supported yet by HyperGraph.

Note: If you want to run ABF converter without using Altair scripts, you can either copy ABF converter in working directory or set ABF_PATH environment variable.

Engine Messages

/FAIL/HASHIN - Clarifications of the rupture message.

Back to top

Highlights of the OptiStruct 12.0 Release

The 12.0 release of OptiStruct is yet another functionality-rich release. This release features an alternative algorithm for topology optimization among other major developments, and significant enhancements were made in the handling and treatment of composites at the free sizing optimization phase. Additionally, the coverage of internal responses was further increased to include multiple new DRESP1 response types. Some key highlights of the 12.0 release are listed below.

Topology Optimization

- An alternative algorithm based on the Level Set Method is available for topology optimization (Beta functionality)

Composites Free Size Optimization

- Ply drop off constraints have been added for optimization of laminate composites
- Manufacturing constraints for laminate composites can now be defined locally, and BYPLY and BYSET in addition to orientation. This is available for free sizing and sizing optimization
- OUTPUT,FSTOSZ (option to automatically generate a sizing model after free sizing of a composite structure) has been rewritten using a new pure ply based approach

Free Shape Optimization

- Automatic re-meshing is available for free shape optimization (Beta functionality)
- The smoothing algorithms have been significantly improved

Internal Responses (DRESP1)

- Equivalent radiated power (ERP) is available for NVH requirements
- Equivalent plastic strain is supported in nonlinear response optimization

Contents

12.0 Release Notes - Topology Optimization

Thermal and Thermo-mechanical Subcases

Level Set Method

Move Limit Controls

12.0 Release Notes - Composite Free Size and General Free Size Optimization

Manufacturing Constraints in Composites Optimization

Automatic Offset Updating

Output Control

Designable Core Layer

Multiple Properties Reference on the DSIZE Card

12.0 Release Notes - Free Shape Optimization

Automatic Re-meshing

Mesh Smoothing

12.0 Release Notes - Internal Responses (DRESP1)

- Equivalent Radiated Power
- Equivalent Plastic Strain
- Mechanical and Thermal Strains for Composites
- Eigenvector
- Stress Constraints for a Cluster of Elements

12.0 Release Notes - Output

- DVCREL Data
- Shape Variables with Heat Transfer Subcases
- Design Property Output
- MINMAX/MAXMIN Reporting
- Optimization History Output
- Assemblies in H3D

12.0 Release Notes - Performance

- Parallelized Solutions
- Efficient Approximations

12.0 Release Notes - General Enhancements

- Support for Unsupported BLOCK Data in the BULK Data Input Format Solution for Nonlinear Response Optimization
- New ANALYSIS Option on the DESVARG Card
- Sequential Quadratic Programming (SQP) Algorithm
- DGRID (VARTYPE# on DRESP2 Card)
- DDVOPT,2 with SHAPEOPT,2
- Long Data Format for the .grid File
- LTID and UTID for DRESP2 and DRESP3

12.0 Release Notes - Improved Error Handling and Messaging

12.0 Release Notes - Resolved Issues

Introduction

Altair OptiStruct is the state-of-the-art finite element based structural optimization tool for a wide variety of engineering problems based on linear statics and linear dynamics, as well as complex nonlinear transient dynamics, and multi-body dynamics for mechanical systems.

Back to top

12.0 Release Notes - Topology Optimization

Thermal and Thermo-mechanical Subcases

Topology optimization is supported for thermal and thermo-mechanical subcases. Conduction properties are appropriately accounted for during optimization.

Level Set Method

An alternative algorithm based on the Level Set Method is available for topology optimization. The Level Set Method for topology optimization will produce clearly defined structures with sharp boundaries and no checker boarding in cases where the loading is dependent on the optimized structure (for example gravity loading and thermal boundary conditions). This is being released as Beta functionality in version 12.0.

Move Limit Controls

Move limit controls have been implemented for topology optimization with nonlinear responses via the nonlinear geometry solution, i.e. with the equivalent static load method. This improves the robustness of the optimization solution.

Back to top

12.0 Release Notes - Composite Free Size and General Free Size Optimization

Manufacturing Constraints in Composites Optimization

- Ply drop off constraints have been added for optimization of laminate composites. The directional thickness gradient for a laminate can be specified as a design constraint for the design of composite layups
- Manufacturing constraints for laminate composites can now be defined locally, and BYPLY and BYSET in addition to orientation. This is available for free sizing and sizing optimization
- The core, non-design plies, or both, can be excluded from the LAMTHK, PLYTHK and PLYPCT manufacturing constraints
- Free size optimization with pattern grouping is now available for components manufactured from Tailor rolled sheet metal blanks

---

Automatic Offset Updating

For composite or shell models with ZOFFS or Z0 defined on the elements or properties, the values are updated based on the thickness values as they change during composite free sizing or sizing

optimization. This allows the TOP or BOTTOM surface of the shell to remain constant during the optimization process as the thickness changes.

Output Control

OUTPUT,FSTOSZ (option to automatically generate a sizing model after free sizing of a composite structure) has been rewritten using a new pure ply based approach so that sub-stacks and interfaces are supported. The advanced algorithm can be used for output of 2-, 4- and 8-ply bundles, and advanced control can be used to ignore elements in a given ply orientation when the thickness is less than 5% of the maximum thickness. Additionally, elements are deleted when they do not belong to any of the ply bundles, and SMCORE is properly handled in the FSTOSZ output.

With OUTPUT,FSTOSZ the thickness of ply bundles can be multiples of the manufacturable ply thickness.

Designable Core Layer

The core layer is now designable when used with the SMCORE laminate option. A specific option for use with BYPLY is available on the CONST entry to retain the core as non-design.

Multiple Properties Reference on the DSIZE Card

Previously, mixed ply structures were not supported. The DSIZE card can now reference multiple properties to support various modeling scenarios.

Back to top

12.0 Release Notes - Free Shape Optimization

Automatic Re-meshing

Automatic re-meshing is available for free shape optimization. During free shape optimization if the mesh becomes too distorted it will be automatically re-meshed. 2D and 3D re-meshing is supported. This is being released as Beta functionality in version 12.0.

Mesh Smoothing

- A new NTRANS option has been added to the DSHAPE card to smoothen the transition from design grids to non-design grids for free shape optimization.
- The smoothing algorithm for free shape optimization has been significantly improved.
- An improved algorithm is now available for free shape on curved shells when SMETHOD=2 (alternative mesh smoothing algorithm) is defined.

Back to top

12.0 Release Notes - Internal Responses (DRESP1)

Equivalent Radiated Power

- Equivalent Radiated Power (ERP) is available as a response for all optimization types.

Equivalent Plastic Strain

Equivalent plastic strain is available as a direct response for nonlinear response optimization using the equivalent static load method. This is supported for NLGEOM, IMPDYN and EXPDYN subcase types.

Mechanical and Thermal Strains for Composites

In the past only total strain was supported as an internal response. Now, the total, mechanical only, or thermal strain only can be used as responses.

Eigenvector

Components of the eigenvector are now available as internal responses. This allows matching and targeting mode shapes.

Stress Constraints for a Cluster of Elements

Stress constraints for a cluster of elements can now be defined. This provides a way of removing artificial stress concentrations from the formulation, which would otherwise have an influence on the optimization solution.

Back to top

12.0 Release Notes - Output

DVCREL Data

Optimized element data controlled by the DVCREL data are output to the `.prop` file.

Shape Variables with Heat Transfer Subcases

Shape design variables are output for optimization with heat transfer subcases so that analysis results can be plotted on the updated shapes.

Design Property Output

OUTPUT,PROPERTY has been enhanced to support “ANY” and “DESIGN” as arguments.

OUTPUT,PROPERTY,ANY specifies that all properties should be output (including non-design properties), while OUTPUT,PROPERTY,DESIGN specifies only designable properties be output.

“DESIGN” is the default behavior.

MINMAX/MAXMIN Reporting

Previously, the reported MINMAX/MAXMIN value was always the value of the first objective function (whether it was retained or not), instead of being the actual minimum/maximum value. The output has been enhanced by looping through the retained objectives and selecting the appropriate minimum/maximum value.

Optimization History Output

The HyperGraph ASCII results file, .hgdata is now output for nonlinear response optimization.

Assemblies in H3D

Assembly structures are supported in the H3D output for analysis, optimization, FlexH3D and H3DDMIG.

Back to top

12.0 Release Notes - Performance

Parallelized Solutions

The nonlinear solution can be run in parallel while performing a nonlinear response optimization.

Efficient Approximations

Efficient approximations have been implemented for large scale size optimization type of problems, i.e. problems with many design variables and many stress type responses. This is applicable for static problems and enables running large models in much shorter times and with much less memory. Stress and strain responses for shells, composites and 1D elements are supported. Efficient approximations can be used through the DOPTPRM,APPROX,REDUCED data.

Back to top

12.0 Release Notes - General Enhancements

Support for Unsupported BLOCK Data in the BULK Data Input Format Solution for Nonlinear Response Optimization

The ASSIGN command has been enhanced to support two new file types; viz.; Starter and Engine file types, for nonlinear analysis based on Bulk data input run via the nonlinear geometry solution. These files are specified using ASSIGN,STARTER and ASSIGN,ENGINE control card options. Simple modifications are allowed in the internally generated Block data file, using the content in these files through INSERT and REPLACE. This is useful for advanced nonlinear response optimization studies.

New ANALYSIS Option on the DESVARG Card

This allows an optimization to be started with the design variable properties being initialized based on the corresponding analysis properties.

Sequential Quadratic Programming (SQP) Algorithm

Sequential quadratic programming (SQP) has been made available in version 12.0. This is an alternative primal method typically used for size and shape optimization types of problems. SQP can be activated by using the DOPTPRM,OPTMETH,SQP data.

DGRID (VARTYPE# on DRESP2 Card)

Grid location in a DRESP2 function is now based on the local coordinate system by default. DGRIDB can be used to specify grid location in the basic coordinate system.

DDVOPT,2 with SHAPEOPT,2

DDVOPT,2 (a continuous optimization phase followed by a discrete optimization phase starting from the continuous optimum) is now automatically activated when SHAPEOPT,2 (alternate algorithm for shape optimization) is used along with discrete design variables.

Long Data Format for the .grid File

The long data format can be used for increased accuracy and can be written out using the PARAM,GRIDFORM,LONG data.

LTID and UTID for DRESP2 and DRESP3

LTID and UTID (table identification number of a TABLEDi entry that specifies the lower and upper bound as function of a loading frequency) are now supported for DRESP2 and DRESP3 responses on the DCONSTR and DOBJREF cards.

Back to top

12.0 Release Notes - Improved Error Handling and Messaging

- OptiStruct errors out when TMINPLY and the designable ply thickness are the same. TMINPLY should be less than the designable ply thickness.
- For responses used in modal frequency response optimization, the ATTB field on the DRESP1 card cannot be blank when used with FREQ3, FREQ4 or FREQ5 loading frequency cards, and when the number of modes on the EIGRL card is not specified. A clear error message is printed if this is the case.
- OptiStruct appropriately errors out when the normal velocity of grids referenced as responses, cannot be calculated. This may occur due to the grids not being connected to any shell or solid elements, or not being located on the surface of the model.
- Improved error message when an unreasonably large number of potential starting points is used with the global search option.
- Improved error message when multiple sub-ranges are defined for the same response for DCONSTR or DOBJREF.
- Improved checking for manufacturing constraints that cannot be satisfied because they do not involve designable (or existent) plies, for composites optimization.

Back to top

12.0 Release Notes - Resolved Issues

- A global search option run would incorrectly use the design property values when run with the –analysis run option. This issue has been fixed and the analysis property values are now used when run with the –analysis run option
- When a frequency sub-range was used with DCONSTR or DOBJREF, the convergence criteria would incorrectly take into account inactive responses that were not within the range of frequency. This has been resolved and the active responses within the specified frequency sub-range are considered for convergence

- REPSUB, REPGLB, DREPORT, etc. are now skipped when the -optskip run option is used
- Topology optimization with no manufacturing constraints or member size control can now be combined with composite free sizing optimization
- Non-design plies are now correctly accounted for in the volume fraction calculation
- The penalty is now correctly ignored for mass and inertia calculations in free sizing optimization of shells
- Nodal thicknesses are now accounted for in formulations involving stress/strain constraints
- Element groups (EG#) defining zones for zone-based free-sizing optimization are now correctly read in the solver as detailed in the documentation for GROUP, the optional continuation line for zone-based free-sizing optimization, on the DSIZE card
- The order of DSIZE cards defined in the input file – with GROUPs for zone based free sizing – can now be defined differently from the order of STACK cards in the input file
- Convergence has been improved and made more robust in the equivalent static load implementation for nonlinear response optimization
- Second order tetrahedral elements can be generated from the standalone version of OSSmooth

Back to top

Highlights of the AcuSolve 12.0 Release

- AcuSolve V12.0 boasts massive improvements to parallel scalability when running the solver on high core counts.
- AcuSolve V12.0 features a number of improvements targeted at handling the largest of customer models. AcuPrep has been enhanced to handle models in excess of 1 billion elements, and domain decompositions as high as 131,072 subdomains.
- A new acoustic post-processing program, AcuFwh is included in the V12.0 distribution. AcuFwh is used as a post processor for AcuSolve data to propagate the acoustic sources to far field microphone locations that are located outside of the simulation domain.
- AcuTrace and AcuSolve can now be run in a bi-directionally coupled fashion to provide more efficient particle tracing simulations as well as enhanced capabilities.
- AcuConsole no longer needs a separate license feature to import Parasolid files.
- AcuSolve V12.0 boasts a new, common installation package interface across all supported platforms.
- Users now have access to the AcuSolve HTML reference manuals from within the AcuSolve installation.
- Improvements to AcuConsole database robustness
- Added Plug-in and Group Managers to streamline workflow in AcuConsole
- Enhancements to porosity modeling options to support cylindrical and spherical coordinate systems
- Simplified specification of turbulence quantities at inlets
- New utility for mapping results between meshes
- New visualization capabilities in AcuConsole
- Modifications to advanced solution strategy settings in AcuConsole to improve database performance
- A number of changes have been made to the supported platforms and third party packages in the V12.0 release.

Contents

Drastic Improvements to Parallel Scalability

Increased Capability for Massively Parallel Applications

Introduction of Ffowcs-Williams-Hawkings Acoustic Solver

Bi-directional Coupling Support Between AcuSolve and AcuTrace

Changes to AcuConsole Parasolid CAD Reader

Improvements to Installation Package

Updated and Integrated AcuSolve Reference Manuals into the HyperWorks Help System

Usability Improvements

Changes to Supported Platforms and Third Party Packages

Notable Resolved Issues

Summary of New/changed AcuSolve Input Commands
Summary of New/Changed Command Line Options
Summary of Commonly Encountered Known Issues

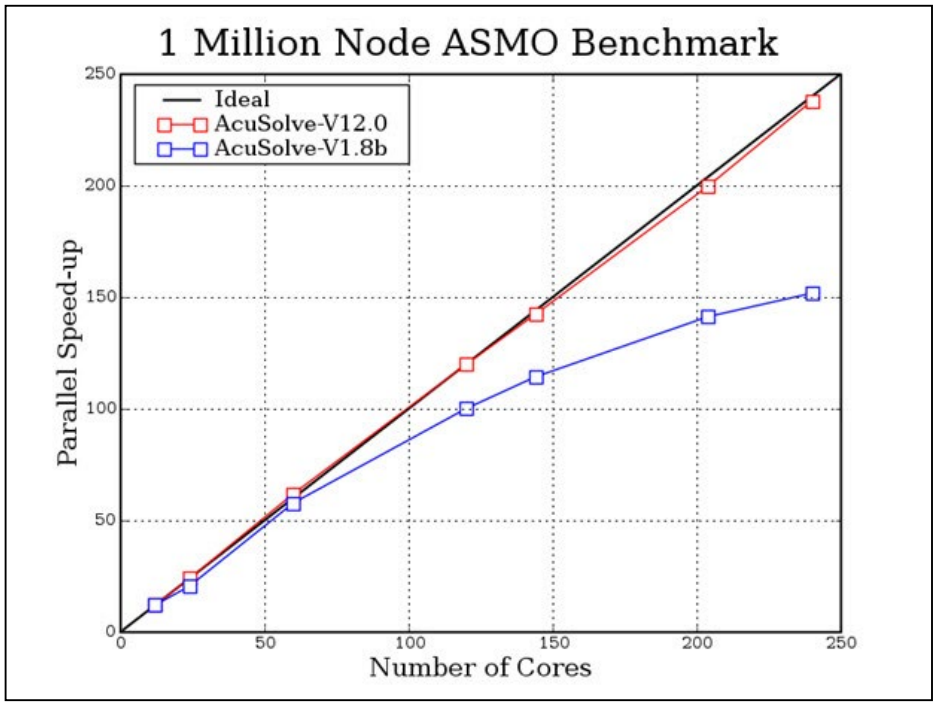
Introduction

The V12.0 release of the AcuSolve product suite comes equipped with many new features, improvements, and resolved issues. This release continues to build on our class leading CFD solution by providing performance improvements for massively parallel applications, introducing new acoustic capabilities, and providing even tighter integration into the HyperWorks software suite.

12.0 Release Notes

Drastic Improvements to Parallel Scalability

AcuSolve V12.0 boasts massive improvements to parallel scalability when running the solver on high core counts. In past releases of AcuSolve, the general guideline for maintaining linear parallel scalability was achieved by maintaining approximately 10% or fewer of the total nodes on a domain decomposition boundary (i.e. interface node fraction of .1 or less). With AcuSolve V12.0, linear scaling is achieved for interface node fractions reaching as high as .2 in some cases (i.e. 20% of nodes falling on domain boundaries). This improvement results in a net increase in solver speed for nearly all parallel applications, and allows customers to utilize large computer resources more efficiently. The plot of parallel scalability shows representative results indicating the performance improvements between V1.8b and V12.0. Note that the improvements shown were obtained on an intentionally small automotive external aerodynamics model. At the highest core count, each subdomain only contains approximately 4,200 finite element nodes. Larger model sizes will scale to larger core counts. The following table compares results from other common AcuSolve application areas and again reflects the improvements in parallel performance between V1.8b and V12.0.



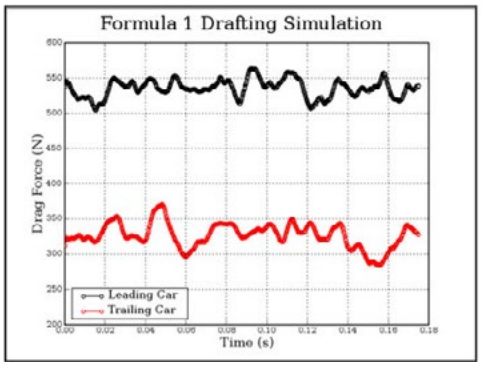
Case Description	Number of Subdomains	Finite Element Nodes	Interface Node Fraction	Speed-up vs. V1.8b
NACA0012 Airfoil - Steady	72	97,636	0.173	1.82x
Road Sign - ALE	240	1,818,405	0.156	1.37x
Wind Turbine Rotor - Steady	240	4,571,720	0.136	1.66x
Wind Turbine FSI - Steady	240	7,593,941	0.128	1.59x
Rotor – Sliding Mesh	120	4,983,066	0.122	1.22x
F1 Race Car - Steady	240	4,803,366	0.113	1.44x
Sedan Model - Unsteady	240	10,876,592	0.091	1.17x

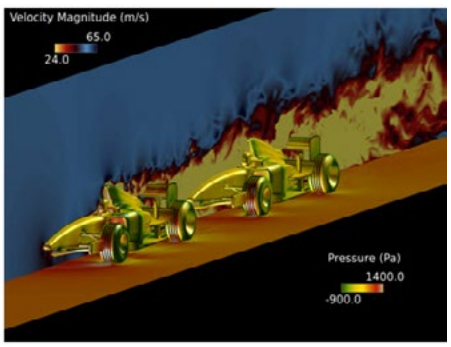
The improvements to parallel scalability are achieved using a combination of technologies in both the solver and pre-processing phase of the solution procedure. Note that the numbers shown above are representative of common applications. Actual speed improvements and scalability will vary with the geometry, mesh, computer platform, and physics being simulated.

Increased Capability for Massively Parallel Applications

AcuSolve V12.0 features a number of improvements targeted at handling the largest of customer models. AcuPrep has been enhanced to handle models in excess of 1 billion elements, and domain decompositions as high as 131,072 subdomains. Combined with the improvements to parallel scalability, these enhancements make AcuSolve the ideal solution for customers seeking to push the limits of simulation technology! The following output shows results for an aerodynamics analysis of 2 notional Formula 1 race cars traveling at approximately 125 MPH. The largest simulation run to date of this configuration is a 1.1 billion element transient analysis using the DDES turbulence model. The plot shows the reduction in drag of the trailing car induced by the wake on the leading car. A visualization of the flow field indicates the highly turbulent wake ingested by the second car and clearly illustrates the cause for reduction in drag.

APPROVED
acuPrep:
Minimum coordinates = -5.7164e+00 -1.6131e-01
Maximum coordinates = 5.7164e+00 5.5047e+00
Total nodes = 169984316
Total elements = 1007704126
Element aggl. = 15
Total aggl. rad. surfaces = 0
Total interface surfaces = 0
Average elements per node = 2.371287e+01
Average element neighbors = 3.991256e+00
Allocated element neighbors = 4.000000e+00
Input CPU/Elapse time = 4.000000e-01 1.796394e+00 Sec
Proc. CPU/Elapse time = 4.301530e+03 4.004976e+03 Sec
Solar CPU/Elapse time = 0.000000e+00 0.000000e+00 Sec
DDC CPU/Elapse time = 8.003840e+03 8.778923e+03 Sec
Output CPU/Elapse time = 5.353780e+03 6.978941e+03 Sec
Total CPU/Elapse time = 1.765955e+04 2.064464e+04 Sec
Total Memory Size = 1.469753e+05 MbytesacuRun:

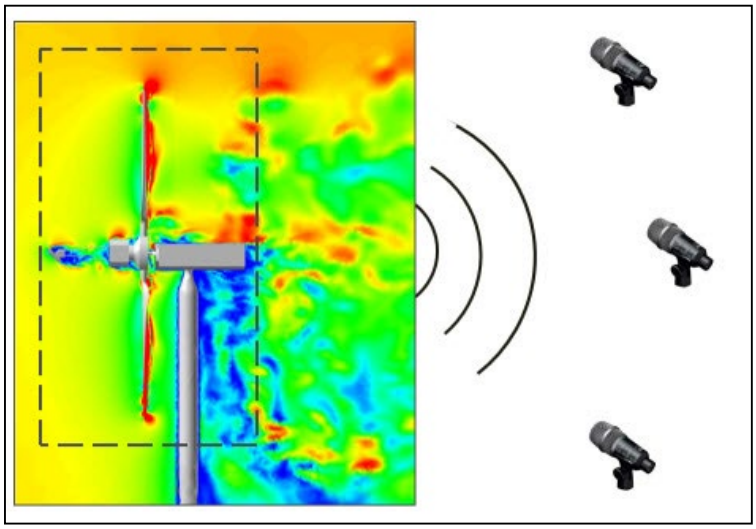




AcuSolve simulation of two F1 cars

Introduction of Ffowcs-Williams-Hawkings Acoustic Solver

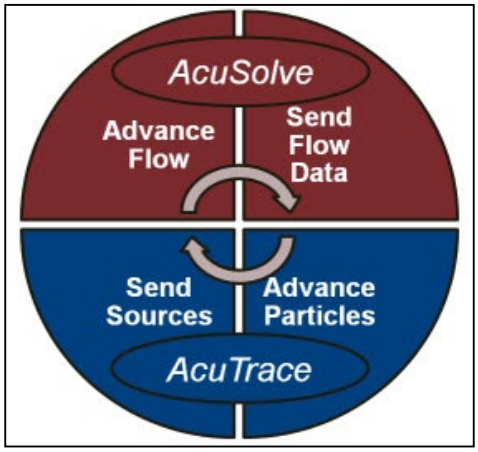
A new acoustic post-processing program, AcuFwh is included in the V12.0 distribution. AcuFwh is used as a post processor for AcuSolve data to propagate the acoustic sources to far field microphone locations that are located outside of the simulation domain. This technology enables users to evaluate the far field acoustic signature of designs using AcuSolve and AcuFwh. At the current time, acuFwh operates as a post processor to AcuSolve and relies on data written to disk using the FWH_SURFACE and FWH_OUTPUT input file commands. AcuFwh is fully parallelized and can be run in shared and distributed memory computing environments. Note that this marks the initial release of AcuFwh, and there is currently no graphical user interface support through AcuConsole. Users should refer to the AcuSolve Command Reference Manual and AcuSolve Programs Reference Manual for guidance on using the new capability.



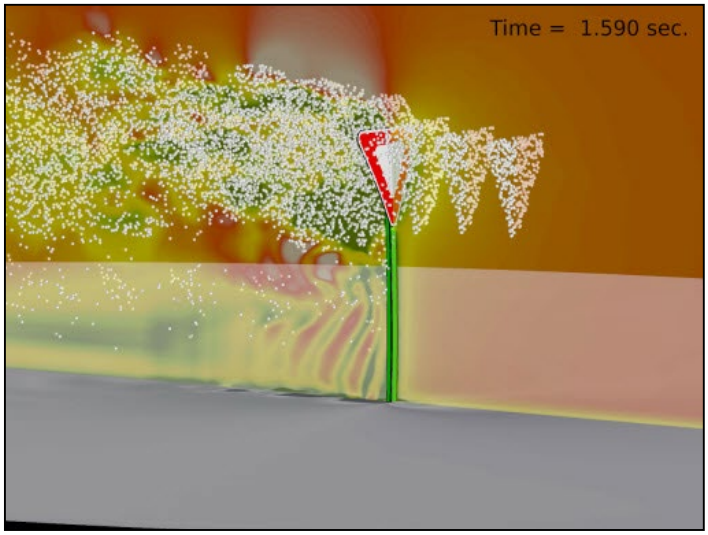
Schematic representation of acoustic source propagation using AcuFwh and AcuSolve.

Bi-directional Coupling Support Between AcuSolve and AcuTrace

AcuTrace and AcuSolve can now be run in a bi-directionally coupled fashion to provide more efficient particle tracing simulations as well as enhanced capabilities. When exercising the bi-directional coupling between AcuSolve and AcuTrace, the two codes exchange information via socket communication. This avoids the need to write nodal output data to disk at high frequencies for later use with AcuTrace. Additionally, this allows AcuTrace and AcuSolve to exchange information regarding the particle properties such as temperature. This enables the simulation of hot/cold particles entering a fluid region and exchanging energy with the surrounding fluid. Additionally, the particles can be used to measure a concentration, which can be used by AcuSolve to determine aggregate material properties for a given element to simulate immiscible fluids.



Schematic representation of AcuTrace/AcuSolve coupling scheme



AcuTrace results for massless particles flowing over a flexible road sign under high wind conditions

---

Changes to AcuConsole Parasolid CAD Reader

AcuConsole no longer needs a separate license feature to import Parasolid files. Parasolid formats may now be imported directly into AcuConsole using the standard HWU licensing scheme.

Improvements to Installation Package

AcuSolve V12.0 boasts a new, common installation package interface across all supported platforms. This was accomplished through a transition to the InstallAnywhere packaging. The InstallAnywhere package is proven to install and uninstall in a fraction of the time on Windows platforms when compared to previous releases of AcuSolve. In addition to the performance improvements, Perl is now included in the Windows packages, eliminating the need for a separate installation to successfully run AcuSolve. Spaces are now fully supported within installation paths and working directory paths on Windows as well.

A single package of AcuSolve is now used to install both the legacy licensed version of the solver (i.e. ACUSIM licensing)as well as the HWU licensed version of the code. This allows all users to take advantage of the improvements to the packaging.

Significant changes have also been made to the directory structure into which AcuSolve is installed. AcuSolve now installs using the following structure: `INSTALL_DIR/altair/acusolve/platform/`. Where platform is one of win32,win64, or linux64 and INSTALL_DIR is chosen by the user at installation time. The $ACUSIM_VERSION variable is no longer used. However, for backward compatibility, it is set to an empty string on Linux platforms.

For users that wish to link the new directory structure within a legacy installation on Linux platforms, this can be accomplished by soft linking the resulting directory structure to the "latest" link, or by soft linking it to a link with the version as the name. Below are some examples:

- HyperWorks and AcuSolve installed in: `/scratch1/Applications/HyperWorks/12.0`
- Legacy AcuSolve installation in : `/scratch1/Applications/Acusim/LINUX64`

From within the legacy AcuSolve installation (/scratch1/Applications/Acusim/LINUX64), the following command should be executed:

```
ln -sf /scratch1/Applications/HyperWorks/12.0/altair/acusolve/linux64 ./V12.0
```

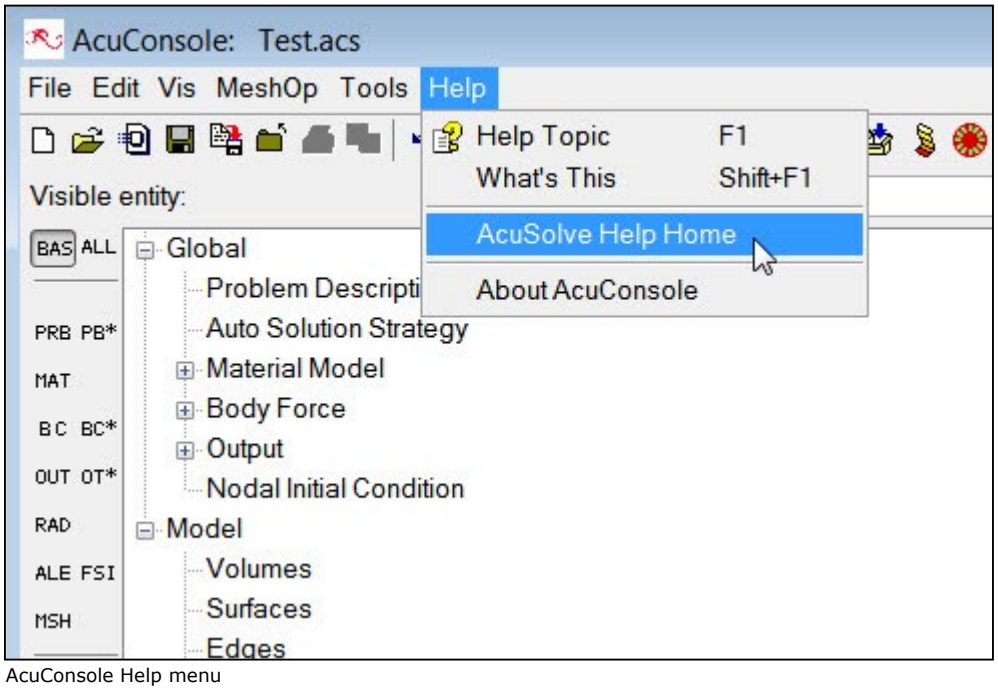
or

```
ln -sf /scratch1/Applications/HyperWorks/12.0/altair/acusolve/linux64 ./latest
```

Changes have also been made to the manner in which the AcuSolve environment is configured on Linux platforms. The necessary shell configuration file names have been modified to `script/acusim.sh` (bourne shell and variants) and `script/acusim.csh` (c-shell and variants), and the structure of them has changed. Installation paths are automatically detected based on the location of the files and the entire distribution directory structure may be moved without the need to modify any files. For backward compatibility, the `acusim.sh` and `acusim.csh` files are soft linked to the filenames of the previous convention. However, users should transition to the new convention as soon as older releases are phased out of use. Note that the automatic detection of the installation path requires the use of the `lsof` command in the csh environment. If this command is not installed on your system, then the `/script/acuCrtcsh` program may be run to hard code the paths in the `acusim.csh` file. This does, however, break the portability of the installation and requires editing of the file if the directory structure is modified.

Updated and Integrated AcuSolve Reference Manuals into the HyperWorks Help System

Users now have access to the AcuSolve HTML reference manuals from within the AcuSolve installation. Upon installing AcuSolve, all manuals are present in the `INSTALL_DIR/altair/acusolve/platform/help/acu` directory. Users can access the reference manuals in a number of ways. A shortcut to the AcuSolve help main page is placed in the AcuConsole **Help** menu on all platforms. On Windows platforms, there is also a link to the AcuSolve Help main page placed in the start menu shortcuts. As a final option, users may also directly load the main `AcuSolveTOC.htm` page using their favorite browser. Note that the PDF reference manuals are also shipped within the `/doc directory` for users who prefer that format.



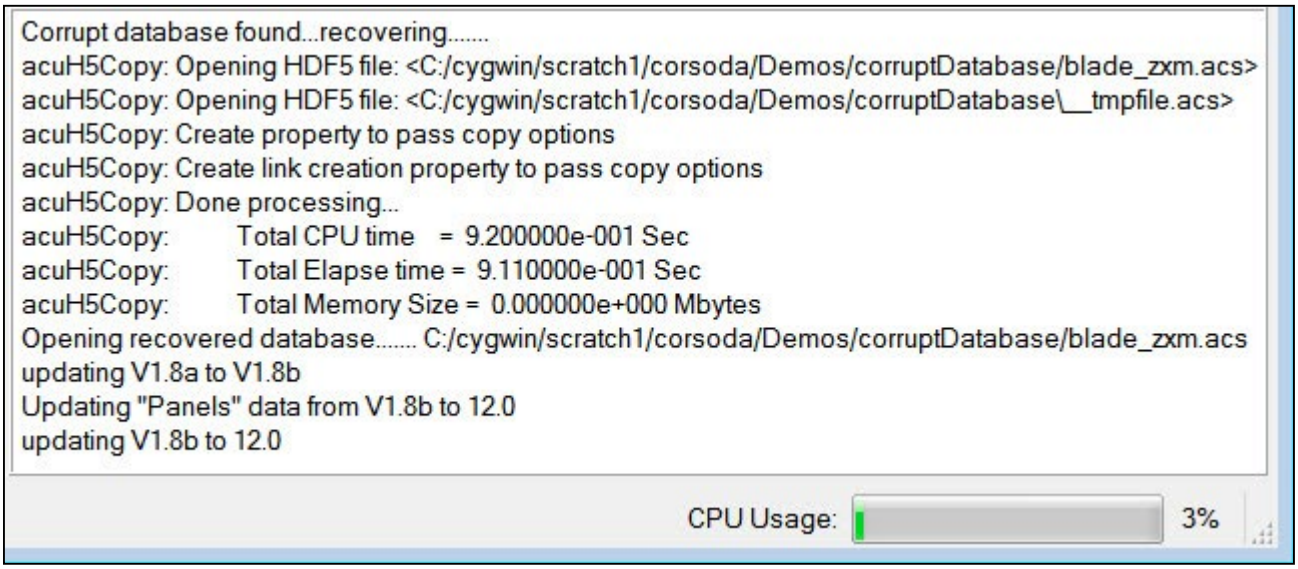
AcuConsole Help menu

Usability Improvements

- **Improvements to AcuConsole database robustness**

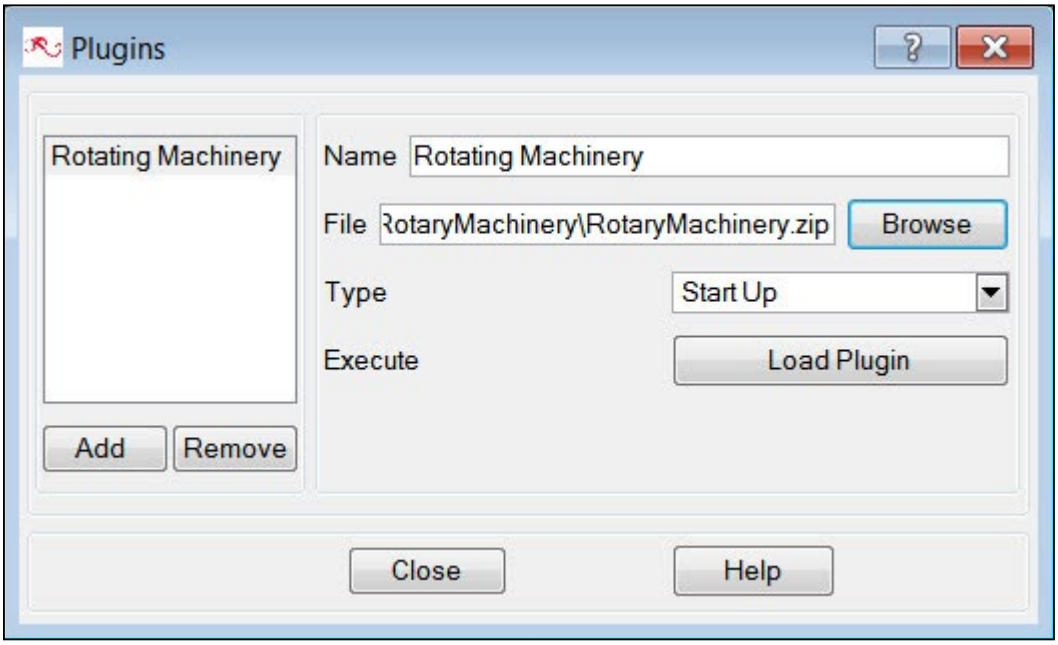
  Two advancements have been made to AcuConsole to address the robustness of user databases.

The primary means of addressing this was through the addition of automatic corruption detection and repair. Upon attempting to open a corrupt database, AcuConsole now detects the problems and makes an attempt to repair the database. Users will see a notification in the message window regarding the database recovery process when this occurs. Once repaired, the database is opened into the current session. In addition to repairing the corrupt databases, AcuConsole has improved control of subprocesses that are launched during the session. Upon exiting AcuConsole, any open sessions of AcuTail and AcuProbe are automatically killed. Unterminated subprocesses upon exiting AcuConsole were a common cause for corruption. Although this improves the situation, it still does not eliminate all possible causes of database corruption.



- **Added Plug-in and Group Managers to Streamline Workflow**

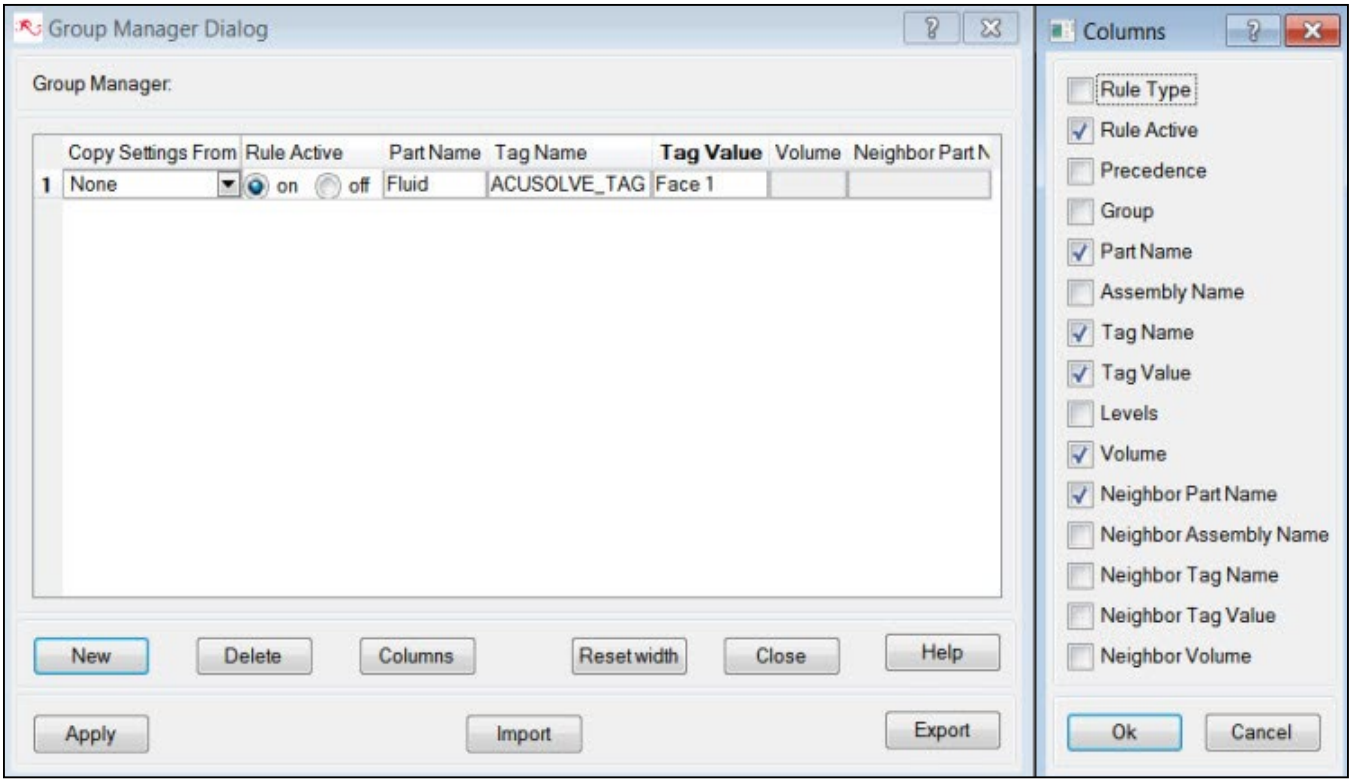
Two new tools have been added into AcuConsole to streamline workflow. For users that have many customization scripts and vertical applications, the Plug-in manager allows users to load their custom python modules into an existing database. Users can select which modules to load, and also specify if they should always be loaded when AcuConsole is launched, or whether they should only be loaded into the current session. Note that this eliminates the need to specify the `-up` option from the command line. Some commonly used plug-ins for working with rotating machinery and external aerodynamics of automobiles are also included in the distribution for users to access. These can be found in the `/plugins` directory.



AcuConsole plug-in manager

The Group Manager provides a new, rule-based method of grouping CAD models. Although AcuConsole has many powerful options for automatically grouping CAD at import time, it was not possible to easily access these tools to regroup geometry for a CAD model that was already in the active session. The Group Manager allow users to define rules, import/export them, and apply them at any time during the pre-processing phase of model construction. Users can define rules based on tag values, surface names, part names, volume names, adjacent component names, etc.

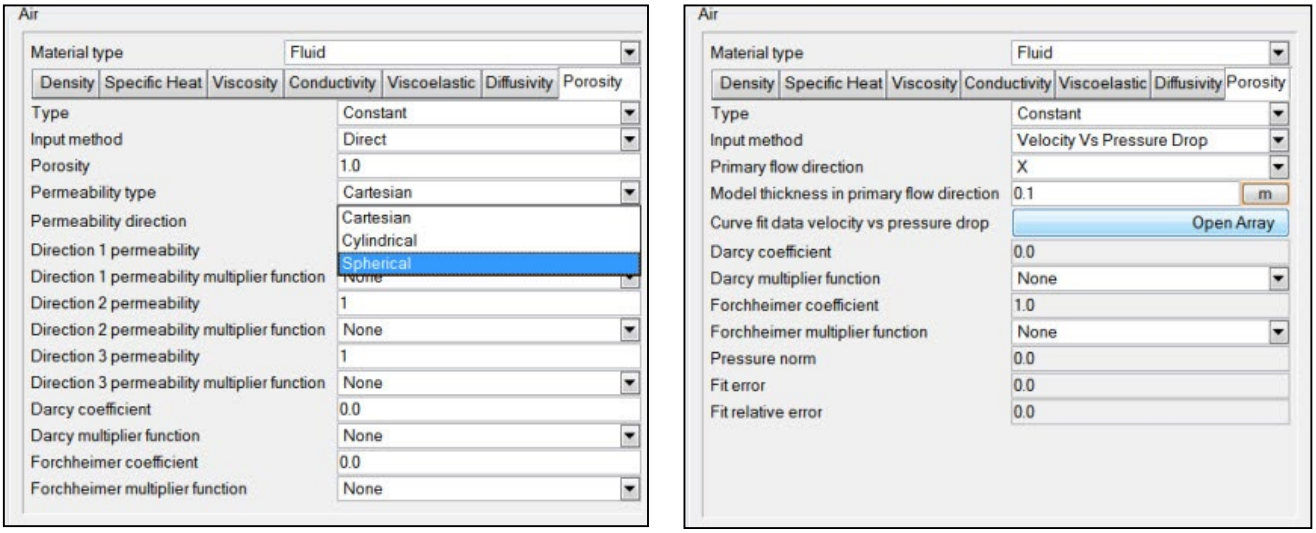
These new features can be accessed through the Tools Menu, or from the Main toolbar in AcuConsole.



AcuConsole Group Manager

- **Enhancements to Porosity Modeling Options**

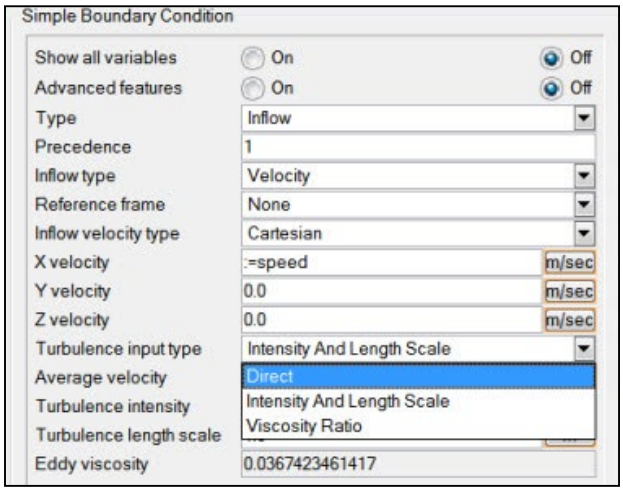
Enhancements have been made to both AcuConsole and AcuSolve to provide more flexible and user friendly porosity modeling options. AcuSolve now supports spherical and cylindrical porosity models. This allows users to specify their porosity model coefficients in non Cartesian coordinate systems to address a broader range of porosity applications. In addition to the new functionality, AcuConsole now supports the specification of the porosity model coefficients through a curve fit of Velocity vs. Pressure drop for Cartesian porosity model types.



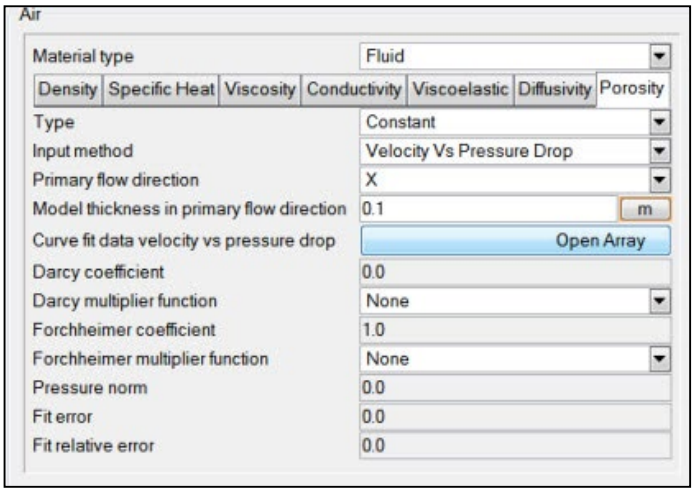
AcuConsole porosity model tab

- **Simplified Specification of Turbulence Quantities at Inlets**

  AcuConsole now supports simplified methods of specifying turbulence model variables at inlet boundaries. When utilizing the Spalart-Allmaras, SST, k-Omega, DES, and SST-DES models, it is necessary to prescribe a suitable level of turbulence intensity at the inflow. When users select inflow types of `flow_rate`, `mass_flux`, and `average_velocity`, these quantities are computed automatically based on an estimated Reynolds Number and distance to the nearest wall. However, when users select `pressure`, `stagnation_pressure` and `velocity` type inflow conditions, it is necessary to specify the appropriate values of eddy viscosity, kinetic energy, and eddy frequency. Options have been added to the inflow boundary condition panel that allow users to select between Intensity and Length Scale or Viscosity Ratio specification for one-equation models, and Intensity and Length Scale or Intensity and Viscosity Ratio for two-equation models. Based on the prescription of these quantities, the appropriate turbulence model variables are computed and applied.



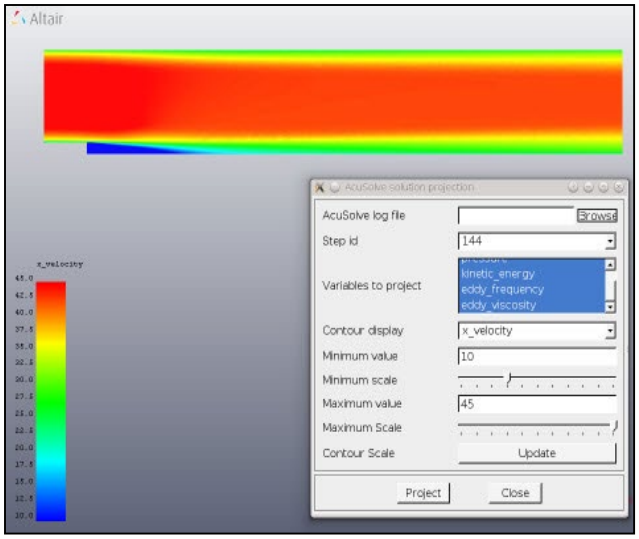
Input options for one-equation models



Input options for two-equation models

- **Utility for Mapping Results Between Meshes**

  Users now have the ability to map results between similar and dissimilar meshes within AcuConsole. This functionality is primarily used for generating initial conditions for simulations. Users can map results from coarse meshes onto fine meshes (or vice versa), or map between similar designs to provide a more suitable initial guess for the solution values. Providing more suitable initial conditions often yields faster convergence of the solver for steady state simulations, and more realistic results at the initial time steps of transient cases. The mapped initial conditions can be visualized on the boundaries of the model using the basic visualization capabilities that are provided in AcuConsole with this utility. This option can be accessed from the ***Tools*** > ***Project Solution*** menu, or from the main toolbar in AcuConsole.

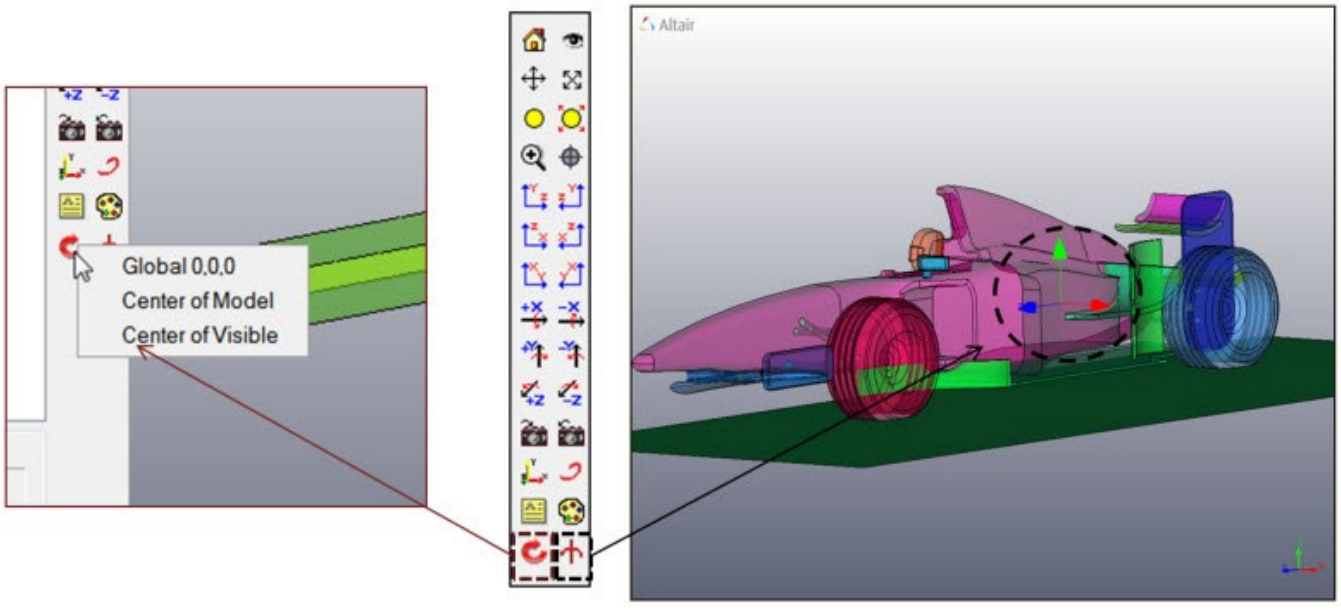


Solutions mapping utility

- **New Visualization Capabilities in AcuConsole**

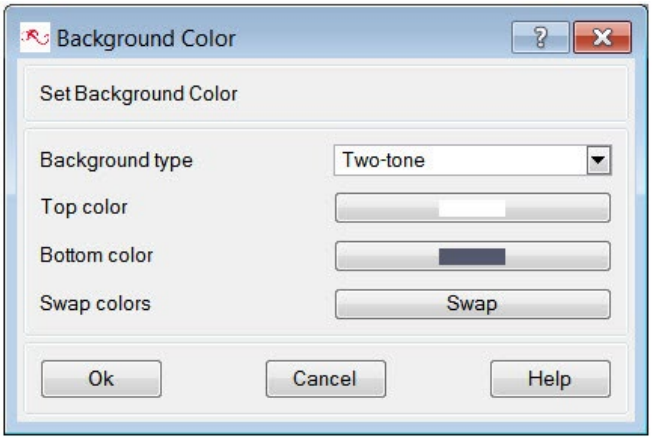
  AcuConsole V12.0 boasts many improvements to the underlying graphics functionality due to a fundamental change in design. Not only does this result in new capabilities delivered to users, but also an overall improvement in frame rates when interacting with complex models.

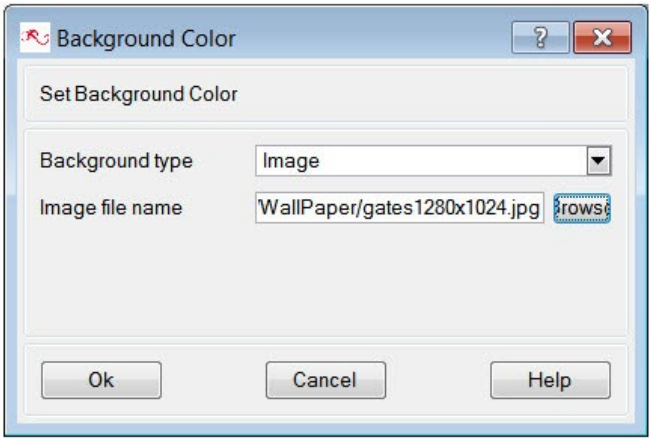
  Among the new capabilities delivered in this release is the ability to modify and visualize spin centers. This new capability allows users to more efficiently manage the view when working with models that are not centered at the origin. Users can select between having their spin center located at the global origin, the center of the model, or the center of the visible entities. The following images illustrate this functionality.

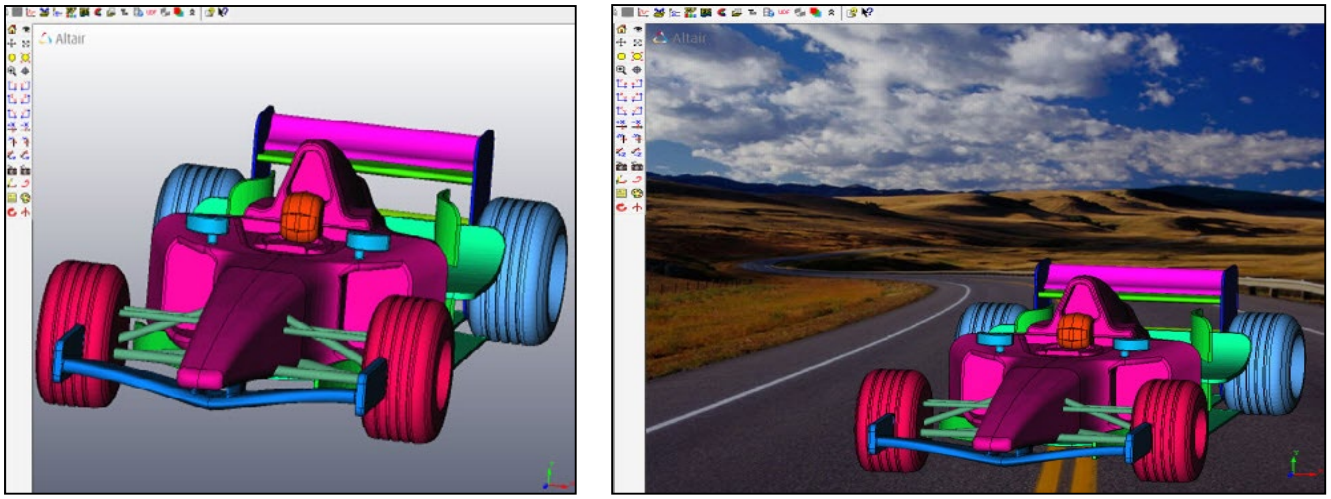


Spin center modification

Users can also select multi-colored backgrounds and utilize pictures as background images in V12.0. These options are available through the standard menus for setting background color. The following images depict the different options.



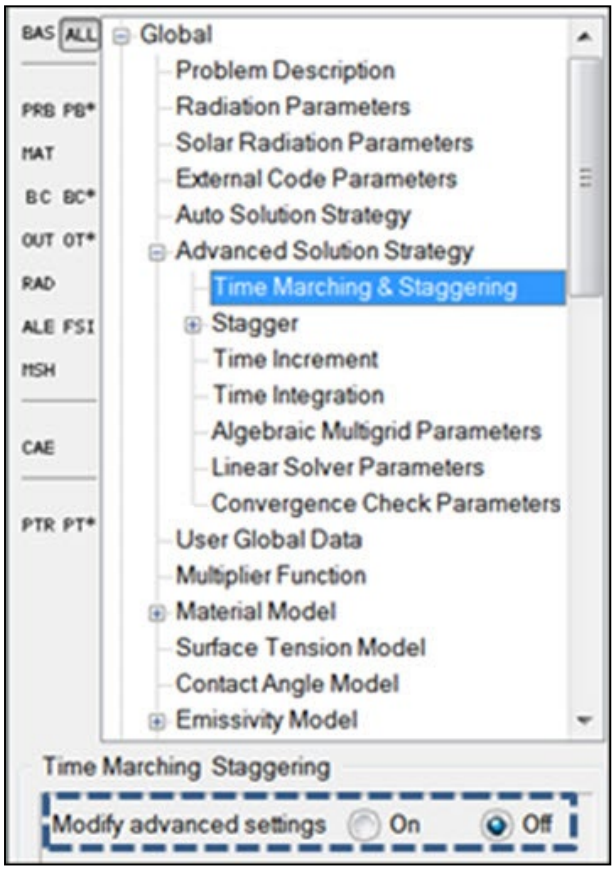


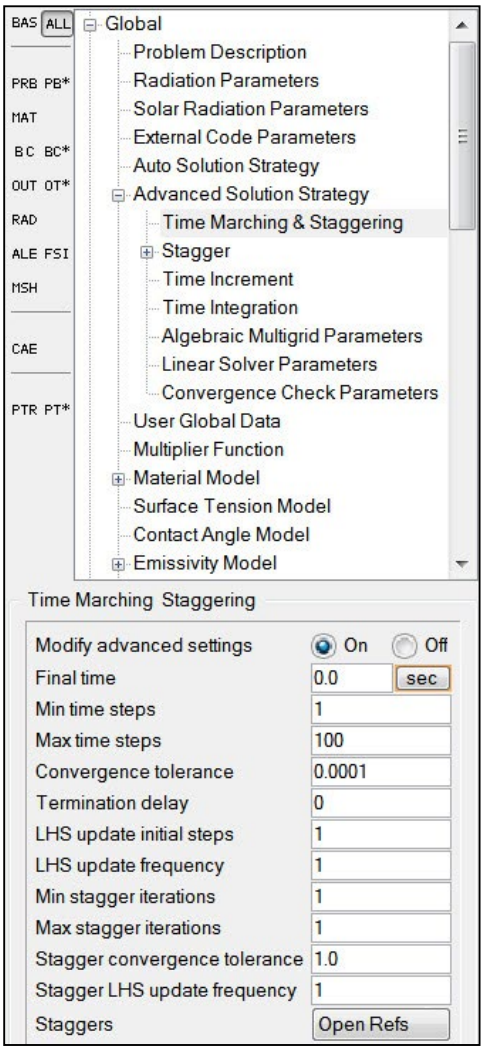


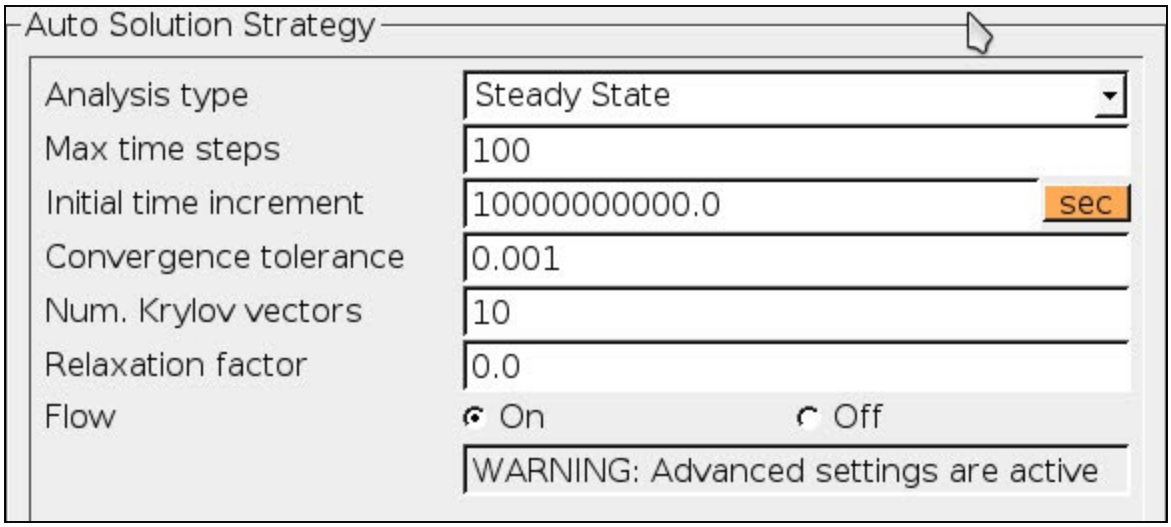
Examples showing different options for background image settings

- **Modifications to Advanced Solution Strategy Settings**

  AcuConsole V12.0 boasts a new approach for setting advanced solution settings. Modifications were made to AcuConsole to simplify the user experience when modifying these advanced parameters, as well as to improve the speed of the database. When users wish to modify the Advanced Solution Strategy settings, they will now need to activate the "Modify advanced settings" button. This will reveal the settings that are available for modification. When this setting is active, all items from that panel are written to the input file and will not be impacted by future changes to the Auto Solution Strategy panel. As a reminder to users, a warning is placed in the Auto Solution Strategy panel when an advanced setting has been modified.







- **Other Notable Changes**

AcuSolve V12.0 contains a number of other notable changes that are worthy of mention. A brief description of each is shown below:

- Added the ability to visualize time history output point locations in AcuConsole.
- Added binary file exchange support between AcuConsole and AcuMeshSim. This option is controlled by the `-ofmt` option of AcuMeshSim and speeds up the reading and writing of meshes to and from disk.
- Improved AcuConsole speed when creating new tree items.
- Improved the speed of AcuConsole's Nastran discrete geometry reader.
- Added ability to skip the import of edges to reduce import time of complex CAD models.
- Added CATIA file import support into AcuConsole.
- Added `cfl_number` to the extended nodal output variable set.
- Added `mass_flux_average` and `bulk_temperature` to surface output variable set. See the AcuTrans entry in the AcuSolve Programs Reference Manual for a definition of these quantities.
- Added `surface_statistics_output` option to the `SURFACE_OUTPUT` command to provide the min/max/standard deviation of surface output nodes. See the AcuTrans entry in the AcuSolve Programs Reference Manual for a definition of these quantities.
- Added a command line option to AcuTrans that removes repeated surfaces from visualizer output files (FieldView, Ensight, H3d, etc.). This option is on by default, and is controlled by the `-rmds` command line parameter.
- Added "coordinates" as an output variable option when writing nodal data to table format.
- Improved PBS integration by adding the `-pbs_attach` option to AcuRun. This option enables better tracking and control of jobs being run through the PBS scheduler.
- Made improvements to the stability and convergence rate of the viscoelastic model.
- Added windowing capability to AcuSpec.
- Added an acufft python module for performing Fast Fourier Transforms.
- Added an `-ismpd` option to the AcuRun command that installs the smpd daemon on Windows systems for Intel MPI and MPICH2.
- Added multiphysics licensing support for AcuSolve/RADIOSS and AcuSolve/AcuTrace co-simulations.

- Added a command line option to toggle between legacy ACUSIM and HWU license types. This options defaults to hwu licensing. Users should set `lm_service_type` = classical for ACUSIM licensing.
- Clarified the naming convention for `back_flow_type`. The following `back_flow_type` options are now available for temperature: `value`, `area_average`, `exiting_area_average`, `bulk`, `exiting_bulk`. For all other scalars, the following options are available: `value`, `area_average`, `exiting_area_average`, `mass_flux_average`, `exiting_mass_flux_average`. Note that the previously available options are still supported, but will no longer be documented. See the `SIMPLE_BOUNDARY_CONDITION` entry in the AcuSolve Command Reference Manual for a definition of these quantities.
- Made Intel mpi the default MPI package on all platforms.
- Officially deprecated the legacy version of AcuTrace and replaced it with AcuTrace2. The old version is now packaged as AcuTraceClassic.

Changes to Supported Platforms and Third Party Packages

A number of changes have been made to the supported platforms and third party packages in the V12.0 release. The most notable changes are listed below:

- Discontinued support for all tools on Linux32 platforms.
- Discontinued shipment of ParaView on all platforms.
- Added Intel MPI support on all platforms.
- Updated to V4.0 of Intel MPI.
- Added support for Cray MPI on Linux64 systems. Note that this only functions on Cray hardware.
- Added the STREAM memory benchmark (AcuStream) to all platforms.
- Added an AcuSolve direct reader to HyperView.
- Updated the JMAG interface version on Windows platforms.
- Updated Abaqus ODB libraries to support reading of ODB V6.12 files.
- Upated AcuFieldView to V13.2.
- Added support for AcuRadTherm to interface results with RadTherm from ThermoAnalytics.

Notable Resolved Issues

- Resolved an issue with DC-FSI simulations that was causing separation of the fluid/structural surfaces.
- Resolved an issue with INTERFACE_SURFACE command that caused gap_factor to cause unpredictable results.
- Fixed a bug in AcuConsole that led to increasing memory footprint when re-importing meshes.
- Fixed a bug in AcuConsole and AcuImport that prevented importing Fluent meshes when quadrilateral surface elements were present.
- Resolved a problem with libz.so.1 that was causing corruption of some Linux desktop environments.
- Fixed a bug in automatic turbulence initialization when no inlets were present in the model.
- Resolved a problem with missing files for Octigabay mpi support.

Summary of New/changed AcuSolve Input Commands

The following table summarizes the changed or newly introduced AcuSolve input file commands. Note that a full description of each command is available in the AcuSolve Command Reference Manual.

Acusolve Command Name	Description
PARTICLE_TRACE	New command to support run time coupling with AcuTrace.
EQUATION	Added upper_convected_maxwell_log_model option to the viscoelastic parameter. Added option to support coupling with AcuTrace.
AUTO_SOLUTION_STRATEGY	Added support for run time coupling with AcuTrace.
STAGGER	Added support for equation=particle to permit run time coupling with AcuTrace.
NODAL_INITIAL_CONDITION	Added support for particle_source variables to permit run time coupling with AcuTrace.
CONVERGENCE_CHECK_PARAMETERS	Modified the default value of viscoelastic_stress_residual_check to be standard.
POROSITY_MODEL	Added permeability_type parameter along with corresponding options to support types of cartesian, cylindrical, and spherical.
PERIODIC_BOUNDARY_CONDITION	Replaced the viscoelastic_xx_stress, viscoelastic_yy_stress, … variables with a single variable named viscoelastic_stress.
SIMPLE_BOUNDARY_CONDITION	Updated naming conventions of all *_back_flow_type options.
SURFACE_OUTPUT	Added support for surface statistics output through the statistics_output_frequency and statistics_output_time_interval parameters.
FWH_SURFACE_OUTPUT	New command to support acoustics post processing with AcuFwh.
FWH_OUTPUT	New command to support acoustics post processing with AcuFwh.

Summary of New/Changed Command Line Options

The following table summarizes the changed or newly introduced command line options for AcuSolve programs. Note that a full description of each option is available in the AcuSolve Programs Reference Manual.

Program	Option	Description
AcuRun	-hosts_list	Added an option of _auto, which sets the value to the local host name.
All programs that support the lm_daemon option.	-lm_daemon	Added and changed the default option to _auto, which sets the value to $ACUSIM_HOME/$ACUSIM_MACHINE/bin/acuLmd.

All programs that support the lm_license_file option.	-lm_license_file	Added and changed the default option to _auto, which sets the value to $ACUSIM_HOME/$ACUSIM_MACHINE/license.dat.
All programs that support the lm_checkout option.	-lm_checkout	Added and changed default option to _auto, which sets the value to $ACUSIM_HOME/$ACUSIM_MACHINE/bin/acuLmco.
All programs that support the lm_server_host option.	-lm_server_host	Added an option of _auto, which sets the value to the local host name. When this value is set to _undefined, HWU licensing is used.
AcuRun	-ismpd	Installs the smpd daemon on Windows machines. Valid only for mp=impi or **mp=mpi** and when running on Windows platforms.
AcuRun	-mp	Added cmpi for Cray machines.
AcuRun	-vmp	Added cmpi for Cray machines.
AcuRun	-pbs_attach	Option to enable tighter integration with PBS schedulers.
AcuRun	-ppn	Number of processors per node to use. Used only when running on Cray machines.
AcuRun	-ppnc	Number of processors per node available. Used only when running on Cray machines.
AcuRun	-numactl	Option to bind processes to cores using numactl command on Linux operating systems.
AcuRun	-mpirun	Added the _system option to run the system level installation of Platform mpi.
AcuRun	-lmtype	Added the hwu option to run using HyperWorks Units licensing.
AcuRun	-tune	When toggled on, this option will run Intel's mpitune application to optimize message passing speed. Only available when using mp=impi.
AcuPrep	-num_solver_threads	When this value is greater than 1, a two level domain decomposition is performed. The first optimizes for the distributed memory message passing, and the second for the shared memory message passing.
AcuView	-lmtype	Added the hwu option to run using HyperWorks Units licensing.
AcuSolve	-lmtype	Added the hwu option to run using HyperWorks Units licensing.
AcuSolve	-exctrace	Option used to export an MPI_TRACE file used for profiling the message passing within a parallel run.

AcuTrans	-rmds	When toggled on, this option removes duplicate surfaces from visualizer output files (Fieldview, H3d, Ensight, etc.).
AcuTrans	-oss	Option to translate surface statistics output.
AcuTrans	-osss	List of surface output sets for which to translate the surface statistics output.
AcuTrans	-ossv	List of surface statistics output variables to translate.
AcuRunTrace	-mp	Added `cmpi` for Cray machines.
AcuRunTrace	-mpirun	Added the `_system` option to run the system level installation of Platform mpi.
AcuRunTrace	-lmtype	Added the `hwu` option to run using HyperWorks Units licensing.
AcuLmg	-code	Added option for acuFwh.
AcuLmg	-lmtype	Added the `hwu` option to run using HyperWorks Units licensing.

Summary of Commonly Encountered Known Issues

The following table summarizes some of the more commonly encountered known issues with AcuSolve V12.0.

Program	Description	Platform
AcuFieldView	AcuFieldView direct reader crashes when loading time step 0 if user has `output_initial_condition = on`.	win32, win64
AcuFieldView	AcuFieldView direct reader crashes when loading data that was generated with `collect_subdomain_output = on`.	win32, win64
AcuConsole	A Python segmentation fault occurs upon exiting AcuConsole after a database has been recovered. This error is harmless and does not impact the user's data.	All
AcuConsole	After scaling a model using Model > "Transform Coordinates", the right-click menu "Info" function does not reflect the updated coordinates when displaying the max/min/center of a surface or volume group.	All

Back to top

Highlights of the HyperXtrude Solver 12.0 Release

HyperXtrude for Metal Extrusion

- Built-in bearing optimization solution
- Improvements to bearing friction model
- Superalloy extrusion using glass lubrication
- Starter Billets with different lengths and temperatures

HyperXtrude for Polymer Processing

- Improvements to heat transfer prediction
- Improvements to profile deformation prediction

HyperXtrude for Metal Rolling

- Support for both symmetric and full models
- Export loads to do stress analysis with RADIOSS
- Support for user defined material models and results

HyperXtrude for Friction Stir Welding

- Export loads to do stress analysis with RADIOSS
- Support for user defined material models and results

HyperXtrude for Billet Forging

- A new solver to simulate billet forging

HyperXtrude for Resin Transfer Molding

- A new solver to simulate resin transfer molding

Contents

12.0 Release Notes - HyperXtrude for Metal Extrusion

Hensel-Spittel Constitutive Model

Particle Traces using AcuPT

Profile Shape Factor

Indirect Extrusion Results

Tool Deflection Analysis

Bearing Optimization

Indirect Extrusion – Moving/Fixed Die

Billet Data and Starter Billet

Tube Extrusion – Glass Pad

Strain Rate Data in Results

User Defined Functions

Mass/Volume Statistics

Output Filenames
Resolved Issues

12.0 Release Notes - HyperXtrude for Polymer Processing

Heat Transfer Predictions
Tool Deflection Analysis
Profile Deformation
Resolved Issues

12.0 Release Notes - HyperXtrude for Metal Rolling

Hensel-Spittel Constitutive Model
Symmetric and Full Models
Improvements to Slab Deformation
Tool Deflection Analysis
User Defined Functions
Output Filenames
Resolved Issues

12.0 Release Notes - HyperXtrude for Friction Stir Welding

Hensel-Spittel Constitutive Model
Tool Deflection Analysis
User Defined Functions
Output Filenames
Resolved Issues

12.0 Release Notes - HyperXtrude for Billet Forging

Hot and Cold Forging
Round to Round Forging
Square to Square Forging
Grain Size and Other Results

12.0 Release Notes - HyperXtrude for Resin Transfer Molding

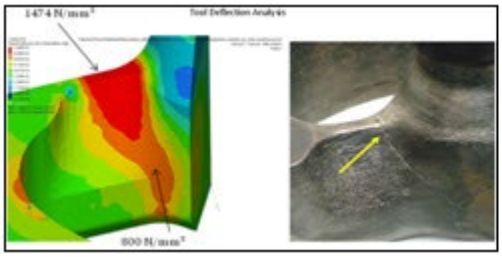
Two Phase Filling Model
Vacuum Assisted RTM
Heat Transfer Analysis
Curing Kinetics
Resin Flow Front and Fill Time Prediction

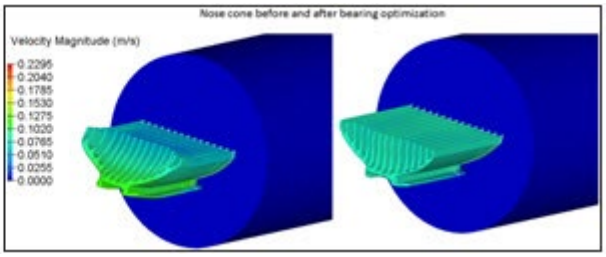
Introduction

Altair HyperXtrude is a finite element solver for simulation of metal extrusion, polymer extrusion, metal rolling, and friction stir welding processes. It has process specific features for each of these processes.

Back to top

12.0 Release Notes - HyperXtrude for Metal Extrusion





Hensel-Spittel Constitutive Model

Hensel-Spittel model is implemented in HyperXtrude for modeling the workpiece. The Hensel-Spittel model describes the flow stress as function of temperature, strain rate, and strain. Including the dependency on strain improves the accuracy of flow stress calculation. Hence, the predicted results are more accurate and agree well with experiments.

HyperXtrude now has four built-in constitutive models for workpiece:

- Sine Hyperbolic Inverse
- Metal Forming Power-law
- Norton-Hoff
- Hensel-Spittel models

Particle Traces using AcuPT

HyperXtrude can now create particle traces using AcuPT, which is part of AcuSolve software. In this release, this feature is limited to steady state solution. Generated particle trace data will contain a history of velocity, pressure, temperature, and strain for each particle.

Profile Shape Factor

Computes the shape factor and prints this data in an `.out` file.

Indirect Extrusion Results

Indirect extrusion module will now consider the shape factor of the profile in the friction calculations. This enhancement will increase the accuracy of the results.

Tool Deflection Analysis

Support to export pressure loads from the material flow analysis in HMASCII format for use in the Tool Deflection Wizard is implemented.

Bearing Optimization

A built-in bearing profile optimization module with the HyperXtrude solver. This direct and faster approach does not require HyperStudy. This approach groups the bearing control points into design, fixed, and dependent points. By adjusting the design points based on the material flow dynamics, it balances the material flow. Unlike the HyperStudy analysis, you have to only categorize the points; any additional/involved setup is not required.

Indirect Extrusion – Moving/Fixed Die

HyperXtrude will now support both variants of indirect extrusion. In the fixed die scenario, which is the default indirect extrusion, the dummy block and container move at ram speed. In the movie dies case, the die moves at ram speed in the negative extrusion axis.

Billet Data and Starter Billet

You can specify cycle dependent billet data. This includes billet length, ram speed and billet preheat. All these data can vary from cycle to cycle for the specified initial cycles.

Automatic scaling of variable time step data based on the billet length has been implemented. This will allow the data to be specified based on the meshed length of billet and scaled appropriately in each cycle based on the billet length.

Tube Extrusion – Glass Pad

HyperXtrude can model glass pad used for lubrication in extrusion of super-alloys. This setup is supported by the Tube Extrusion Wizard in the user interface.

Strain Rate Data in Results

HyperXtrude will print the strain rate table in the output file when the parameter "WriteStrainRateTable" is set to "yes". In addition, when enabled via H3D Optional results, HyperXtrude will export strain rate tensor to the H3D file.

User Defined Functions

There are few upgrades to this module. The maximum number of user-defined nodal and element results is now increased to 10. In addition to the primary variable, now all the computed data is sent to the UDF routines. These include rate of deformation tensor, temperature gradients, billet interface, skin interface, and stresses.

Mass/Volume Statistics

Print volume and mass statistics of each component (at every time step during transient analysis).

Output Filenames

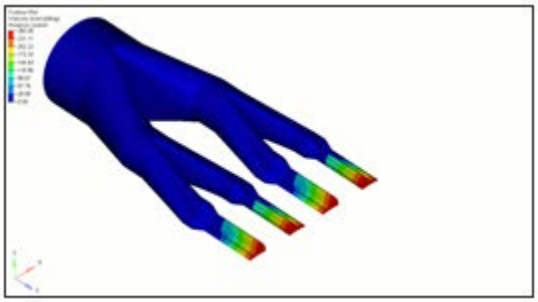
HyperXtrude output file naming convention has changed. Now the current output files will always have the name `<jobname>.<ext>`. If that folder has previously computed results, those files will be renamed.

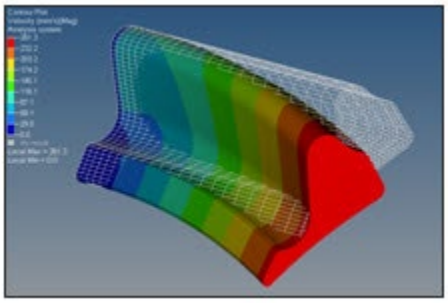
Resolved Issues

- Heat balance table printed in the .out file will include tool surface BC.
- Minor issues in the UDF hooks for material data have been fixed by rigorously comparing all material models implemented via UDF with built-in material models.
- Errors in the unit conversion for strain rate and other data in the UDF module has been fixed.
- Errors in exporting loads (missing loads), has been fixed.
- Bug fixes to error in unit conversion in different modules.
- Bug fixes to creation of data structure in bearing and profile 3D region.
- Bug fixes to choked bearing computation model.
- Improvements to weld strength prediction.
- In bearing optimization, if the initial bearing design is already optimal, solver was not writing the H3D result file. This is fixed and in addition, each new bearing profile is written in a separate file.

Back to top

12.0 Release Notes - HyperXtrude for Polymer Processing





Heat Transfer Predictions

Improvements to solution for models made of non-hex elements with insulated boundary faces.

Tool Deflection Analysis

Support to export pressure loads from the material flow analysis in HMASCII format for use in the Tool Deflection Wizard is implemented.

Profile Deformation

Profile deformation in steady state polymer extrusion analysis is computed using a new algorithm. This method shows how the profile exits the die by accounting for the profile speed together with the deformation in other two directions. In addition, it has parameters to control/scale the computed deformation.

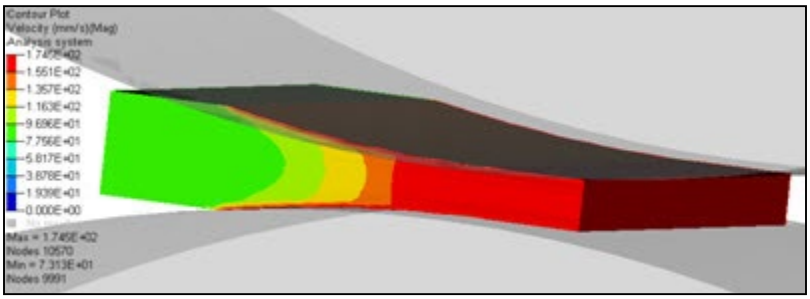
Resolved Issues

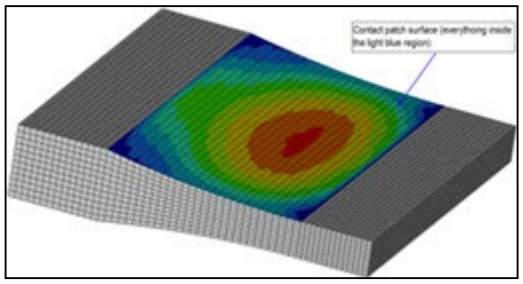
- Improvements to SUPG implementation to handle insulated BCs for non-hexahedral elements.

- Heat balance table printed in the .out file will include tool surface BC.
- Minor issues in the UDF hooks for material data have been fixed by rigorously comparing all material models implemented via UDF with built-in material models.
- Errors in the unit conversion for strain rate and other data in the UDF module has been fixed.
- Errors in exporting loads (missing loads), has been fixed.
- Bug fixes to error in unit conversion in different modules.

Back to top

12.0 Release Notes - HyperXtrude for Metal Rolling





Hensel-Spittel Constitutive Model

Hensel-Spittel model is implemented in HyperXtrude for modeling the workpiece. The Hensel-Spittel model describes the flow stress as function of temperature, strain rate, and strain. Including the dependency on strain improves the accuracy of flow stress calculation. Hence, the predicted results are more accurate and agree well with experiments.

HyperXtrude now has four built-in constitutive models for workpiece:

- Hensel-Spittel models
- Sine Hyperbolic Inverse
- Metal Forming Power-law
- Norton-Hoff

Symmetric and Full Models

Simulation will support both symmetric and full models.

Improvements to Slab Deformation

Slab Deformation is now computed using a new algorithm.

Tool Deflection Analysis

Support to export pressure loads from the material flow analysis in HMASCII format for use in the Tool Deflection Wizard is implemented.

User Defined Functions

There are few upgrades to this module. The maximum number of user-defined nodal and element results is now increased to 10. In addition to the primary variable, now all the computed data is sent to the UDF routines. These include rate of deformation tensor, temperature gradients, billet interface, skin interface, and stresses.

Output Filenames

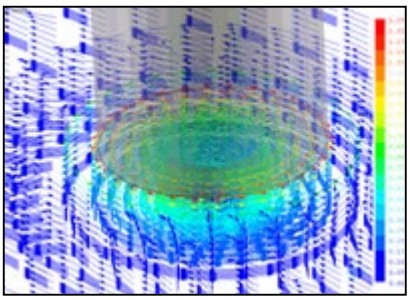
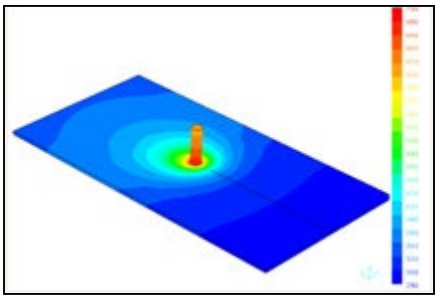
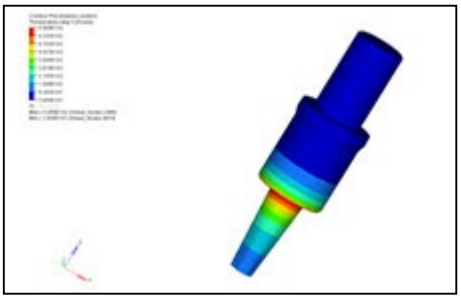
HyperXtrude output file naming convention has changed. Now the current output files will always have the name `<jobname>.<ext>`. If that folder has previously computed results, those files will be renamed.

Resolved Issues

- Heat balance table printed in the `.out` file will include tool surface BC.
- Bug fixes to error in unit conversion in different modules.

Back to top

12.0 Release Notes - HyperXtrude for Friction Stir Welding

Hensel-Spittel Constitutive Model

Hensel-Spittel model is implemented in HyperXtrude for modeling the workpiece. The Hensel-Spittel model describes the flow stress as function of temperature, strain rate, and strain. Including the dependency on strain improves the accuracy of flow stress calculation. Hence, the predicted results are more accurate and agree well with experiments.

HyperXtrude now has four built-in constitutive models for workpiece:

- Sine Hyperbolic Inverse
- Metal Forming Power-law
- Norton-Hoff
- Hensel-Spittel models

Tool Deflection Analysis

Support to export pressure loads from the material flow analysis in HMASCII format for use in the Tool Deflection Wizard is implemented.

User Defined Functions

There are few upgrades to this module. The maximum number of user-defined nodal and element results is now increased to 10. In addition to the primary variable, now all the computed data is sent to the UDF routines. These include rate of deformation tensor, temperature gradients, billet interface, skin interface, and stresses.

Output Filenames

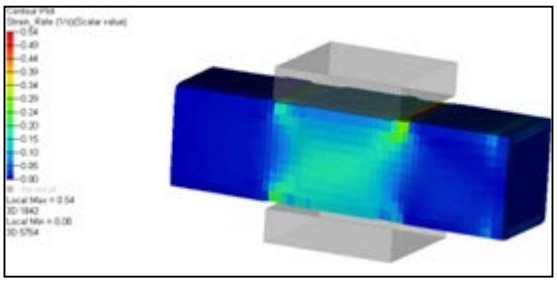
HyperXtrude output file naming convention has changed. Now the current output files will always have the name `<jobname>.<ext>`. If that folder has previously computed results, those files will be renamed.

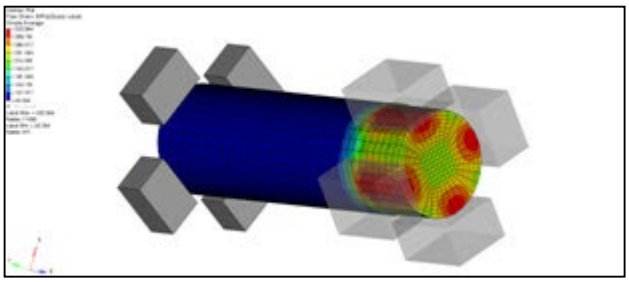
Resolved Issues

- Heat balance table printed in the `.out` file will include tool surface BC.
- Bug fixes to error in unit conversion in different modules.

Back to top

12.0 Release Notes - HyperXtrude for Billet Forging





HX-Forge is a new solver module in HyperXtrude for billet forging simulation. Billets used in extrusion are often forged to obtain a desired cross-sectional shape and desired microstructure (grain size) before being extruded. Hence, this complements the metal extrusion simulation.

Hot and Cold Forging

Support for forging of hot or cold billets.

Round to Round Forging

Support for forging of diameter reduction of circular cross-section billets. Solver will support multiple forging sequences.

Square to Square Forging

Support for forging of square billets. Solver will support multiple forging sequences.

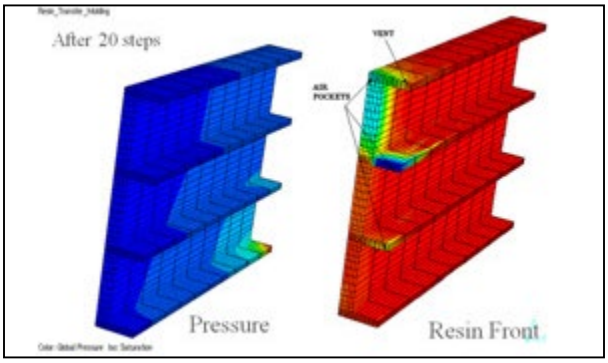
Grain Size and Other Results

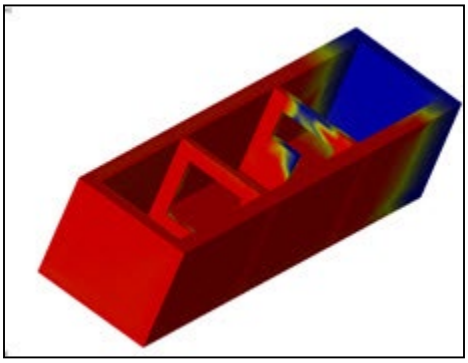
Solver predicts the history of temperature, strain and strain rate during the forging process. Results

also include grain size of the billet and the forces on the punches.

Back to top

12.0 Release Notes - HyperXtrude for Resin Transfer Molding





HX-RTM is a finite element based solver for analysis and design of resin infusion processes, which is used to manufacture polymeric composites.

Two Phase Filling Model

Solver considers both resin (in the filled regions) and air (in the unfilled regions) of the preform.

Vacuum Assisted RTM

Solver uses a weakly compressible model for air to study the effect of vacuum pressure in the filling process.

Heat Transfer Analysis

Solver uses the thermal properties of preform, resin, and air in the heat transfer analysis. Predicted temperature is used in computing the resin viscosity and in curing kinetics.

Curing Kinetics

Curing kinetics is modeled using an nth order and autocatalytic model and is triggered by time delay or temperature.

Resin Flow Front and Fill Time Prediction

Predicted results include animation of resin flow front and fill time. Results include pressure, temperature, degree of cure, velocity, resin saturation, and fill time.

Back to top

Highlights of the HyperForge Solver 12.0 Release

- HX_Forge is a new addition to Altair’s multi-physics offering.

Introduction

HX-Forge is a new solver billet forging simulation. Billets used in extrusion are often forged to obtain a desired cross-sectional shape and desired microstructure (grain size) before being extruded. HX-Forge can simulate round-to-round and square-to-square billet forging. It comes with an easy to use interface (for setting up the model).

Back to top

Highlights of the HyperXtrude-RTM Solver 12.0 Release

- HX-RTM is a new addition to Altair’s multi-physics offering.

Introduction

HX-RTM is a finite element based solver for analysis and design of resin infusion processes, which is used to manufacture polymeric composites. These techniques include Resin Transfer Molding (RTM), Vacuum assisted RTM (VARTM), Zonal Pressure Molding (ZPM) and their variants. HX-RTM is a three-dimensional, transient, two-phase, non-isothermal, and non–Newtonian solver and it includes curing kinetics in the analysis.

Back to top

Highlights of the BasicFEA 12.0 Release

- Basic FEA Accessibility
- New Fatigue Loadstep
- New Meshing Options
- New Basic FEA Toolbar
- Translation
- HTML Report

Contents

11.0.130

Support for Rigid Bodies

Language Support

11.0.120

New User Interface for HyperMesh

Units and Unit Conversion

Material Database

Automatic Contact

Loadstep Types

12.0 Release Notes

Basic FEA Accessibility

- BasicFEA is now accessible through HyperWorks Desktop.
- Split screen for Pre and Post is available while in the BasicFEA environment.

New Fatigue Loadstep

- BasicFEA Materials Database is updated with fatigue material properties.
- EN and SN formulations are available.
- One or two loadstep events are available

New Meshing Options

Hexa/Tetra Elements

- With this option, if the geometry is easily meshed, BasicFEA will produce hexa elements. If the geometry is complex, BasicFEA will default to tetra elements

First/Second Order Elements

- This option controls the mesh and the quality of the results. It can also affect run time.

New Basic FEA Toolbar

BasicFEA process is now captured in the top-down toolbar.

Translation

Translated into new Languages

- Korean, Turkish, Chinese (Simplified), and Chinese (Traditional).

HTML Report

An HTML report is available after a successful run through the **View report** option in the BasicFEA browser right click menu.

Back to top

11.0.130 Release Notes - Basic FEA

Support for Rigid Bodies

- Rigid element connection [RBE2]
- Pin type connection used in a sliding analysis [RBE2 with CGAP]

Language Support

- Located under the Preferences menu
- Available for English, German, French, Spanish, and Japanese

Back to top

11.0.120 Release Notes - Basic FEA

New User Interface for HyperMesh

The objective for this project is to define a user profile that will allow users from the novice to expert level to run a broad range of simple analyses [with RADIOSS (Bulk Format) as the solver in the background].

Moreover we aim to bring the power of our solver to a wider user base using a streamlined process based approach.

Units and Unit Conversion

BasicFEA has four sets of unit systems to choose from, and when you switch systems everything gets converted.

On import BasicFEA reads the unit system associated with the geometry and changes the environment settings. There are simple visual tools that help examine the dimensions of your geometry, so users can scale the model and ultimately choose a proper working unit system.

Material Database

Out of the box, BasicFEA offers 14 common materials to choose from. User materials can also be created and stored for future use, even in a new session.

All the materials are converted between unit systems.

Automatic Contact

A simple searching algorithm will find parts that contact one another and assign them all a tied contact. The option to switch each contact definition to sliding exists in the right-click menu.

Loadstep Types

Linear static, normal modes, buckling and sliding contact will be the available solutions sequences [all solved in RADIOSS (Bulk Format)] for the first release of BasicFEA.

Back to top

Highlights of the Collaboration Tools 12.0 Release

- Recycle Bin for Deleted Contents
- Check in - Replace
- Enhanced Team Support
- Editor Enhancements
- MotionView Integration
- Preference Support within Collaboration Tools Profiles
- Performance Improvements

Contents

11.0.130 Release Notes

Import/Export of Library Contents

Query by Date, User, Comment

Get without References

Selective File Download from PLM

Publish to PLM

User Interface Enhancements

11.0.120 Release Notes

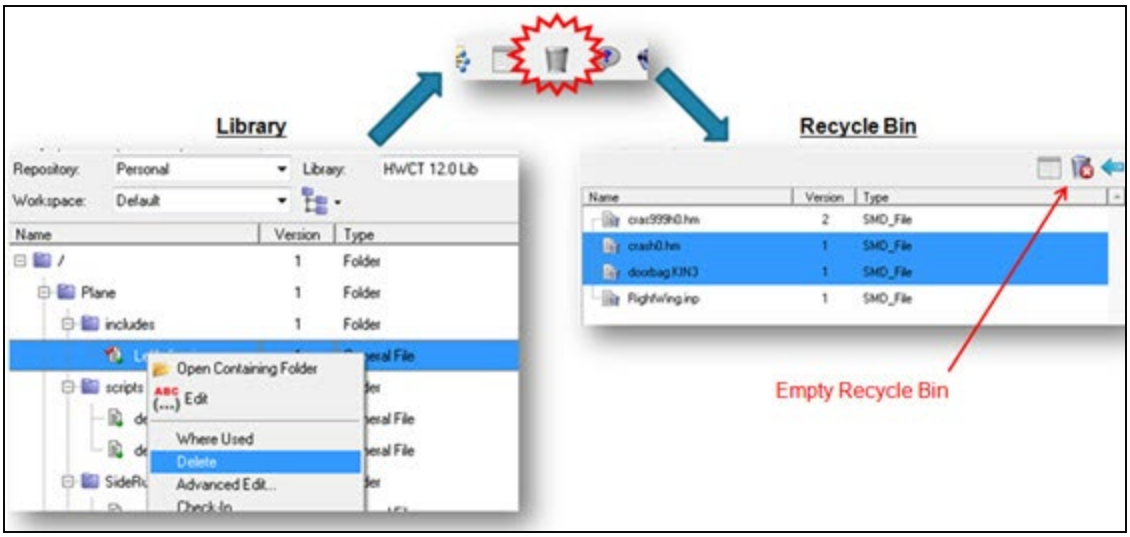
Maintain Traceability

PLM Connector Enhancements

12.0 Release Notes

Recycle Bin for Deleted Contents

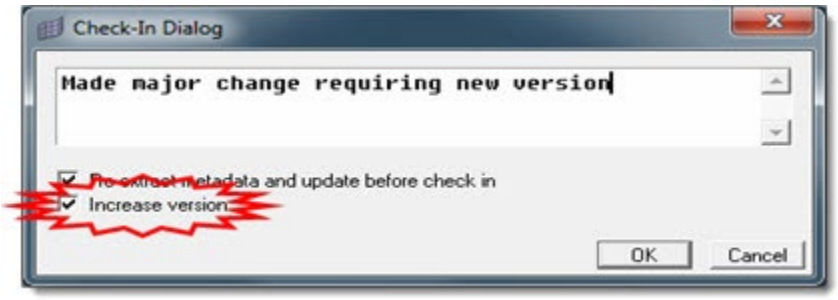
This feature behaves similarly to the recycle bin on the computer's operating system. Contents deleted from a library will move to the new recycle bin. Accessed from the Collaboration Tools toolbar, users can see which contents have been deleted, restore contents to the library, or permanently obliterate them.



By accessing a content's version history, users can now also delete individual versions of a content.

Check in - Replace

With this release, users will be able to decide whether to increment a content's version when checking in the content. Previously, a content's version was incremented with every check in in order to maintain a full version history and pedigree for each content. To provide more flexibility, and to allow users some additional control over content versioning, users may now decide whether to increment a content's version at check in.



Note: This feature is controlled based on two user preferences. One preference determines whether the "increment/don't increment" option will appear on the check in dialog. The other preference sets the default value regarding whether or not a content is versioned upon check in. Depending on a particular customer's policy, the ability to version or not version may not be exposed to users.

To maintain full pedigree of contents, it is recommended that contents be versioned at each check in.

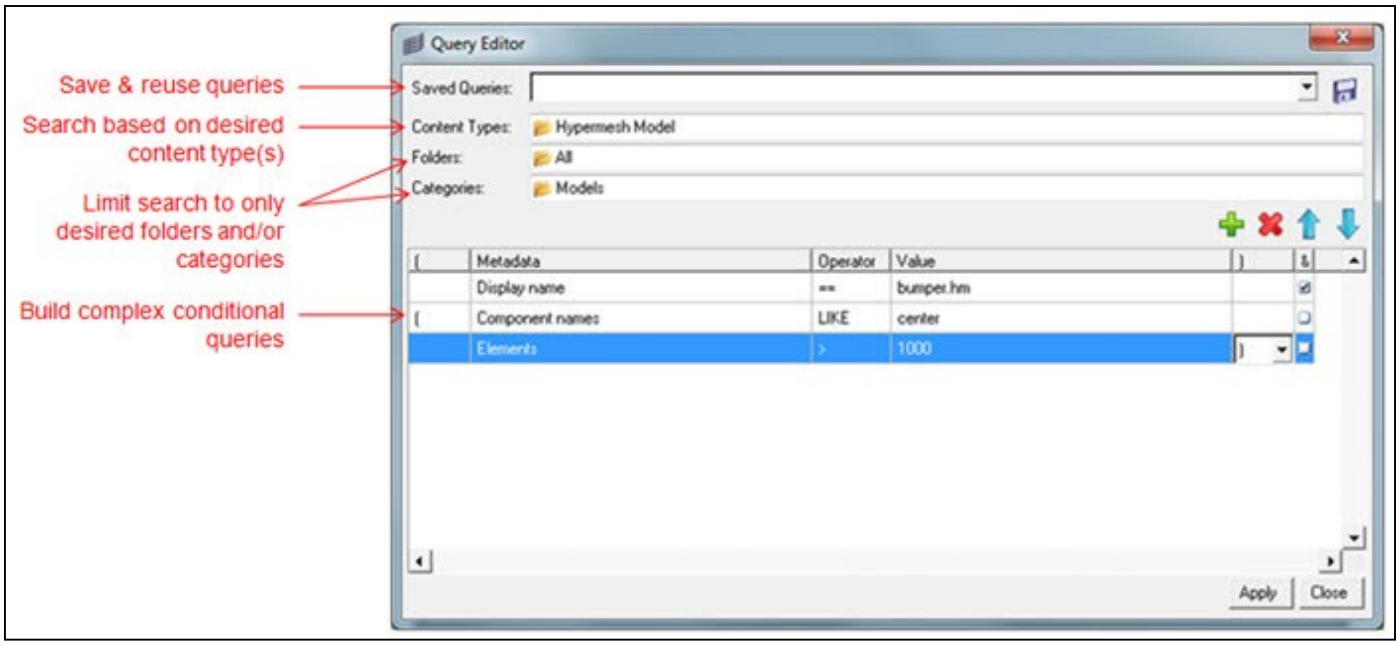
Enhanced Team Support

In order to support larger groups of users, it is now possible for customers to leverage the HyperWorks Enterprise Foundation to maintain a central library.

Note that HyperWorks Enterprise Foundation requires additional installation and configuration in order to deploy.

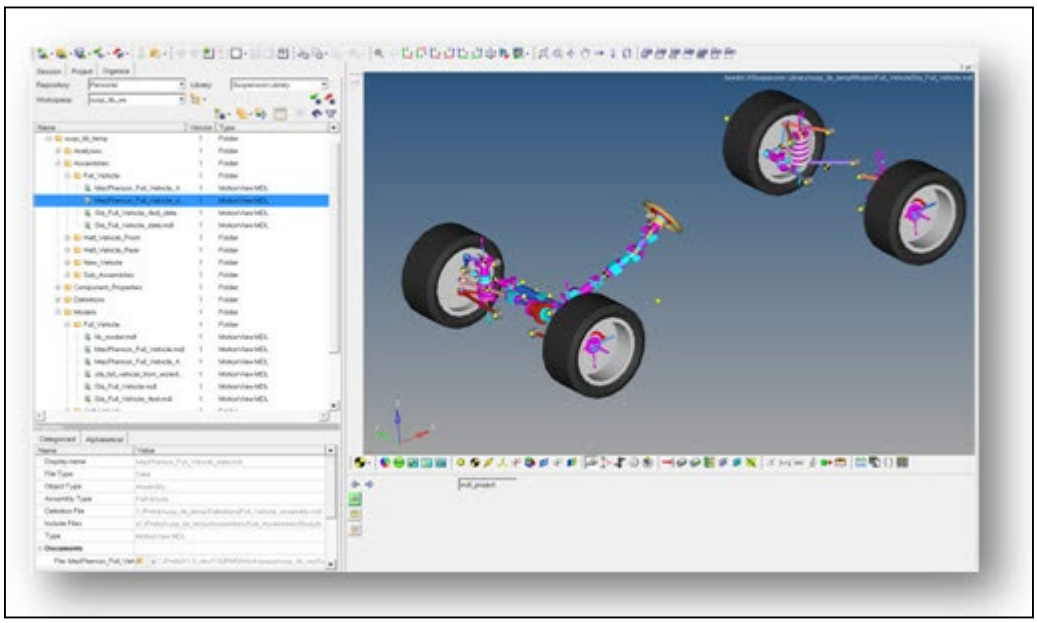
Query Editor Enhancements

The Query Editor for advanced queries has been enhanced for this release. It is now possible to limit queries to only desired folders and/or categories.



MotionView Integration

MotionView users now have Collaboration Tools support within the MotionView client, including the ability to manage MDL files, results files, and other MBD-related date and files.



Preference Support within Collaboration Tools Profiles

It is now possible to include library-specific preferences within Collaboration Tools Profiles.

Profiles are used by customizers of Collaboration Tools to be able to provide customer-based enhancements to meet the specific needs of a particular implementation. They include custom or enhanced content definitions, custom context menus and toolbar options, and other custom behaviors.

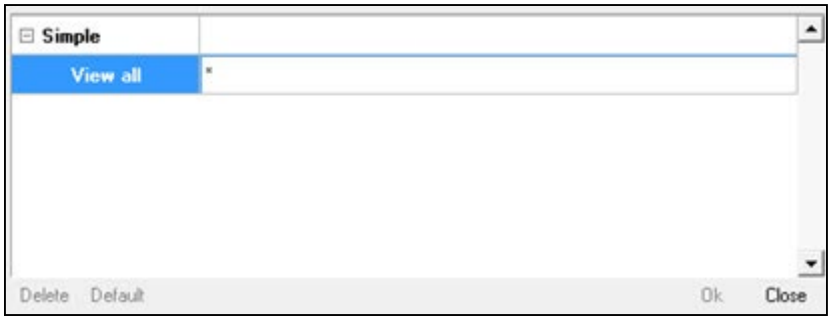
Performance Improvements

With previous versions of the Collaboration Tools, opening a library resulted in all contents of that

library being retrieved and displayed. While convenient, because the user didn't have to perform any queries to see and browse the contents of the opened library, this can lead to reduced performance when opening libraries containing large numbers of contents.

To enhance performance, a default query will be executed when libraries are opened, potentially limiting the number of contents retrieved and improving performance.

To remain consistent with previous releases, an out-of-the-box "View all" default query will be created for each library. However, users can save and set their own default queries in order to limit the number contents retrieved and thus improve performance.



Notes: Create and save a query for all contents ('*') in a library and set that to the default query if you want to search for and display all contents of a library as in the out-of-the-box behavior of past versions.

Back to top

11.0.130 Release Notes

Import/Export of Library Contents

This feature allows for greater portability of Collaboration Tools contents and facilitates greater sharing of information and collaboration between users or groups of users. It also facilitates transfers of information between groups or even organizations in the case of OEM-Supplier data exchange.

- Select contents from a library and export them to a new library which can then be independently added and modified by other users.
- Updated libraries can then be imported back to the original library. A convenient preview dialog allows users to see which contents have been updated, in either library, before importing.

Query by Date, User, Comment

It is now possible to search for contents in libraries based upon when they were created or the last time they were modified. It is also possible to search based upon which user made the last modification and also comments added during the last modification.

Get without References

Users now have the option to retrieve contents to their workspace without retrieving all contents and files referenced by the selected content. Previously, referenced contents and files were always retrieved.

Selective File Download from PLM

It is now possible to selectively choose which files are downloaded when retrieving CAD from PLM.

Previously, when a PLM item was chosen, all its associated files were downloaded. This often resulted in the user being forced to download more information than desired at the cost of performance and efficiency.

Publish to PLM

Users can now upload contents from Collaboration Tools libraries directly into their PLM system. By using the same connect platform used to seamlessly connect to PLM to retrieve CAD, users can now search for desired items in the PLM system and attach simulation files such as meshes, solver decks, results, and reports.

Note: This feature is not supported on 32-bit Linux operating systems.

User Interface Enhancements

Folders and categories are now color-coded so users can easily distinguish whether the Organize browser is in Workspace or Repository view mode.

Combo box selectors, such as the Library or Workspace selectors, can now be closed simply by clicking anywhere else in the application. Previously, these selectors had to be explicitly closed using the Close button.

Back to top

11.0.120 Release Notes - General

Maintain Traceability

Maintain traceability by viewing where a selected content is referenced is now available. Previous versions of Collaboration Tools only supported viewing of contents referenced by (i.e. used by) a selected content; users can now see all contents which reference a selected content (i.e. where the selected content is used).

PLM Connector Enhancements

The Collaboration Tools PLM connector has been enhanced to allow for browsing of part structures to find desired CAD. Previous versions limited searches to the query results themselves. Users can now expand and browse part structures and select individual or multiple CAD files to retrieve.

Back to top

11.0.101 Release Notes

- Select folders or files to bulk add contents to managed libraries.
- Create and manage labels with new Label dialog.
- Associate groups of contents by label.
- The new Check out Warning ensures that users will be warned when attempting to check out contents currently checked out by others.
- References can now be created to any version of a content, rather than to only the latest version.
- Multi-level references are now displayed in the Organize Browser, rather than only a single level.

- Query enhancements include:
  - Saved named queries for reuse.
  - Set default query to be executed when a library is opened.
  - Query by label
- Perform life cycle operations (Get, Check In, Check Out, Undo Check Out) on multiple contents at the same time.
- Selecting a folder or category performs the selected action on all contents.
- View library contents with registered text editor.
- Register text editor via Collaboration Tools Preferences option.
- Ability to directly create contents in the Collaboration Tools libraries via File > Save As and Export
- Automatic metadata updates via File > Save and Export from HyperMesh and HyperWorks Desktop
- Changes to content and metadata are persistent across HyperMesh and HyperWorks sessions
- All 11.0 libraries will be automatically migrated when opened in HyperWorks 11.0.101

Back to top

HyperMath Solutions 12.0 Release Notes

HyperMath Solutions provide general mathematical tools for math scripting, modeling, study, and optimization.

HyperMath	Mathematical language for numerical computation
HyperStudy	Integrated optimization, DOE, and robust design engine

Highlights of the HyperMath 12.0 Release

- Statistical Math Library Items
- Signal Processing Math Library Items
- Language Items
- HyperWorks Integration
- User Interface

Contents

11.0.130 Release Notes

FFT Based Functions

Deprecated Functions

Enhancements

11.0.120 Release Notes

Enhancements

11.0.101 Release Notes

Enhancements

12.0 Release Notes

Statistical Math Library Items

Statistical math library items: Hypothesis tests

Signal Processing Math Library Items

Signal processing math library items: Faster FFT, Bessel filter

---

Language Items

Language items: Copy on write for matrix data

HyperWorks Integration

HyperWorks Integration: Function registration with HyperGraph directly from the editor

User Interface

User Interface: Split pane editor, combined plots & grids pane

Back to top

11.0.130 Release Notes

FFT Based Functions

All FFT based functions now use the FFTW based algorithm. Outputs are no longer padded to the next power of 2 by default. The effected functions are: Conv, Corr, Coherence, Fft, BlockFft, Frf, BlockFrf, Ifft, PSD, CPSD, BlockPSD, BlockCPSD, SAEFilter.

Deprecated Functions

Several functions have been deprecated and will be removed in a future release. These are:

FftReal, FftImag, FftMag, FftPhase, BlockFftReal, BlockFftImag, BlockFftMag, BlockFftPhase, FrfReal, FrfImag, FrfMag, FrfPhase,BlockFrfReal, BlockFrfImag, BlockFrfMag, BlockFrfPhase, DftReal, DftImag, DftMag, DftPhase, Dft, Idft.

Enhancements

- Enhanced editor outlining, line selection ease and also auto completion turns off inside comments.
- A new and enhanced HyperView plug-in API available. A tutorial has been added to the online help.

Back to top

11.0.120 Release Notes

Enhancements

- Area and Polar plots were added.
- A 2D grid window is available for viewing and editing matrix contents.
- Enhanced memory management.

Back to top

11.0.101 Release Notes

Enhancements

- Both infinity & NAN are supported as data types via keywords Inf & NaN, respectively.
- With 3D plotting, three dimensional line, surface, contour and waterfall plots are now available.
- HyperMath Scripts can now be used as plug-in inside HyperView for custom post-processing.

- HyperMath can now be used to implement custom external readers for HyperGraph.
- The relational and logical operators have been extended to support element by element operations on matrix data.
- Bidirectional data exchange between HyperMath and Tcl is possible.
- Matrix can be written using custom numeric format using FWrite. Multiple delimited text files can be read by FReadText.

Back to top

Highlights of the HyperStudy 12.0 Release

- Significant UI Redesign
- General Enhancements and New Features
- Extended Post Processing
- Connections
- New and Improved Methods

Contents

11.0.101 Release Notes

File Export Options

Evaluation of Main Effects

BackgroundMode Option

Labels

Resolved Issues

12.0 Introduction

HyperStudy enables users to explore, understand and improve their system designs using methods such as design-of-experiments and optimization. HyperStudy generates intelligent variations of the parameters of any system model and reveals relationships between these parameters and the system responses. Using HyperStudy, users can make better decisions and optimize the performance, reliability and robustness of their systems.

HyperWorks 12.0 introduces an enhanced and updated version of HyperStudy. The new version of HyperStudy merges the strengths of the classic HyperStudy along with new simulation technology and an improved user experience.

- The improved Study Explorer guides you through the study process, following the steps a typical engineer goes through to evaluate a design. You are asked to identify inputs and outputs, and then specify what you want to learn about the design under investigation, such as optimal values, trade-off options, or directional guidance.
- Entering, monitoring and reviewing data has been simplified through the use of tables. Tables provide convenient features such as sorting, cutting and pasting, and filtering to help you manipulate and understand your models and results better.
- Study sessions are still saved to an .xml file and new in 12.0, the “DataSet” for each approach in the study is now stored in a self-contained and open relational database. This storage method facilitates faster I/O and more flexible post-processing.
- The study directory structure has been updated for consistency and allows for easier automation.

A directory view of all the files associated with a study is available in the new Directory tab in the Study Explorer pane.

- The Approximation module has been replaced by a Fit Approach
- For experienced HyperStudy users, the transition to 12.0 is seamless, as the overall philosophy of the product remains the same.

Back to top

12.0 Release Notes

Significant UI Redesign

- New Welcome Page that allows easy study and quick start examples access
- Attributes and Data are presented in tables with rich set of features
- Approach steps are simplified and commonized
- Restructure of what questions are asked early in the process
- New and enhanced Evaluation Form capabilities
- New editable run Matrix view
- Use of Multiple Threads for the Evaluate form, offering up to seven dynamically updated views
- Enhanced Function Expression Builder
- Enhanced Message log area
- New Directory View
- New reporting functionality

General Enhancements and New Features

- Directory structure is simplified and commonized
- Archive export/import feature
- System Bounds Check Approach
- Batch mode control of tasks on an Approach by Approach basis
- Multi-Execute (Concurrent running of jobs)
- Automatic updater from previous release
- Any of the Approach Run matrix results can be used to create an Input Matrix and fits can be referenced by any of the Approaches now
- Model evaluation order and dependency checking enhanced

Extended Post Processing

- Consistency between Approaches
- More options on how to plot channels
- Multi plot

- Tight coupling with HyperGraph (Double-click in a chart)
- Color Map
- 3D iteration table
- Richer iteration table
- Stochastics Reliability plot

Connections

- Extended capabilities when selecting inputs and outputs in Excel

---

New and Improved Methods

- Fitting Method: Radial Basis Functions
- Reliability-Based Design Optimization Method:Single Loop Approach
- Multi-Objective Optimization Method: Hybrid Method for Multi-Objective Methods

Back to top

11.0.101 Release Notes

File Export Options

File export options (keep or not keep the Include structure of a model) are now respected by HyperMesh when a model is written in the case of the HyperMesh/HyperStudy connection.

Evaluation of Main Effects

The evaluation of Main effects is improved to handle properly failed runs in a DOE.

BackgroundMode Option

When a study (DOE, OPT, or STO) is executed with the BackgroundMode option, the current preferences are applied to the background process.

Labels

Labels (for variables and responses) used in the study are now used by the post-processing modules. This applies if the Labels are unique. In case of similar/redundant labels, the Varnames are used instead to remove possible confusion.

Resolved Issues

- GA and MOGA Optimization: Ranges for parameters are updated.
- Box-Behnken DOE: Levels are now assigned to each design variable automatically. Whereas before, it was needed to review each variable setting.
- Fixed the issue where the initial value is not one of the discrete values assigned to the design variable.

- Fix for the range of the Beta parameters in the exponential distribution.
- Fixes for the copy of distribution parameters from setup to optimization (SORA and ARSM-SORA).
- Fix the crash when the list of images to be added to the HTML report is cleared.
- Fix the issue (Linux only) where HyperMath did not invoke from HyperStudy when a study (DOE, OPT or STO) is executed with the BackgroundMode option.
- Fix the issue where STUDY_RUN_PATH and STUDY_DIR_PATH were truncated (model folder, e.g. m_1/, was missing).

Back to top

Manufacturing Solutions 12.0 Release Notes

Manufacturing Solutions is a unified environment for manufacturing process simulation, analysis, and design optimization. Use the links below to access the release notes for the following applications:

HyperForm	A unique finite element based sheet metal forming simulation software solution
HyperXtrude	Provides custom tools and easy to use wizards for setting up models to analyze metal extrusion, polymer processing, and metal rolling applications using HyperXtrude solver.
HyperMold	Provides a highly efficient and customized environment for setting up models for injection molding simulation with Moldflow (Midplane, Fusion, and Flow3D) and Moldex3D (Shell and Solid).

Highlights of the HyperForm 12.0 Release

Incremental

- Common (RADIOSS and LS-DYNA)
- RADIOSS Only
- Resolved Issues

Die Module

- Enhancements
- Resolved Issues

Blank Nesting

- Enhancements

RADIOSS One Step

- Resolved Issues

Contents

11.0.130 Release Notes - Incremental

Fast (Fine Mesh) Option

Guide pins/Spacers/Pilots

User Process/Auto Process

Process Setup from Geometry

New Materials

Springback Setup

11.0.130 Release Notes - RADIOSS One Step

Cutting Angle Calculation

New Process Type

11.0.130 Release Notes - Die Module

Unflange Tool

S-section Addendum Creation Panel

Drawbead and Drawbar Creation Panel

Resolved Issues

11.0.120 Release Notes - Incremental

Stopper

Tube HydroForming Setup

Resolved Issues

11.0.120 Release Notes - RADIOSS One Step

Results Initializer

11.0.101 Release Notes - Incremental

User Process/Auto Process

Temperature Output in .sta Files

Forming Contact TYPE 21

Output Animation Files

11.0.101 Release Notes - RADIOSS One Step

Output Stresses

Introduction

HyperForm 12.0 adds significant functionality and enhancement in usability and performance on top of the previous releases. In addition to the standalone product, HyperForm is now available as a client in Hyperworks Desktop.

Back to top

12.0 Release Notes - Incremental

Common (RADIOSS and LS-DYNA)

- Support for multi-stage processes to run in batch mode to leverage servers and clusters
- Support definition of user defined location to store user process setup in the HyperForm Preferences.
- Support of Datum, the reference object used for relative positioning of all other objects.
- Option to store objects position in space as Home position for every tool and ability to return back to home position after autopositioning.
- Performance of springback comparison tool significantly improved
- Ability to pick material created in the session without storing in material database.

RADIOSS Only

- Trimming of solid element blanks
- Seamless integration with HyperStudy for process optimization (available on windows platform)
- Improvement of GAP option under Auto Process for better correlation with physical data
- Support of new materials for thermal forming with solid elements. Also database populated with Boron steel and Magnesium materials
- Support of new materials for forming with solid elements.
- Support of seamless springback option in user process with Automatic option

---

Resolved Issues

- Incremental – RADIOSS
  - Drawbeads created using blank from STA file resulting in solver error in starter because of ID overlap
  - Time step calculation for Fast (Fine mesh) setup not working for Yoshida Material and Thermal using shell elements
  - STA file reading resulting in solver error during starter run.
  - Tool and blank temperature not saved in .hf file
- Incremental - LS-DYNA
  - Error in *CONTROL_IMPLICT_SOLUTION with the variable LSTOL value.
  - Error while defining constraints for springback setup.
  - Error while creating input deck for trimming operation

Back to top

12.0 Release Notes - Die Module

Enhancements

- New binder creation by extension of the part•
- Addendum creation supported with option to consider ribs at corner
- Automatic calculation of offset based on thickness and offset value for punch and blank holder creation.
- Organize option under the User Process to consolidate part, addendum and binder to one component to simplify Incremental setup.
- Improvements in the double attach option
- Drawbar creation for complex lines.

Resolved Issues

- Automatic update of drawbar and drawbead after binder edit.
- Delete part does not remove all children entities.

- Punch Opening Line could not be created for symmetric models.
- Existing addendum not masked/deleted when new addendum is created.

Back to top

12.0 Release Notes - RADIOSS One Step

Enhancements

- Seamless integration with HyperStudy for blank size optimization (available on Windows platform)

Resolved Issues

- Duplicate curves created when importing FLC curve

Back to top

12.0 Release Notes - Blank Nesting

Enhancements

- New option to reflect about X and Y axis to create duplicates
- Option to auto nest by translations of parts around each other
- Option to auto nest by rotation of parts relative to the sheet/ Coil
- Creation of HTML report from the blank nesting tool

Back to top

12.0 Release Notes - Post-processing (HyperView)

Enhancements

- Customized drop down menu for HyperForm including all stamping related features
- Section Cut to plot variation of results along cross-sections
- Draw-in plot to measure movement of blank boundary
- Blank-Tool gap plot to visualize final blank position with respect to the tools

Back to top

11.0.130 Release Notes - Incremental

Fast (Fine Mesh) Option

Fast (Fine mesh) option based on AMS technology in the solver has been introduced to run models with fine mesh but with speed equivalent to the Fast with adaptive mesh option.

Guide pins/Spacers/Pilots

Guide pins/Spacers/Pilots have been introduced in Incremental RADIOSS and LS-DYNA model setup to realistically model the tools that are used to position the blank in the forming process.

User Process/Auto Process

Show/Hide/Isolate functions as part of right click menu under User Process to efficiently review and interact with objects in the model.

The User Process has been updated to support multiple trim lines and local trim direction for each trim line.

Support of sinusoidal tool motion profile under User Process and Auto Process is now available.

Process Setup from Geometry

Support of process setup from geometry for tools and blank is now available.

New Materials

Material for solid elements has been added to the material database.

Springback Setup

Springback setup in RADIOSS has been updated to improve solution robustness and convergence.

Back to top

11.0.130 Release Notes - RADIOSS One Step

Cutting Angle Calculation

Cutting angle calculation has been updated to consider the cutting tool surface profile.

New Process Type

A Process type has been added to one step interface to set advanced options that reflect the forming process characteristics.

Back to top

11.0.130 Release Notes - Die Module

Unflange Tool

Unflange tool has been enhanced to handle unflanging of multiple layers of surfaces.

S-section Addendum Creation Panel

S–section Addendum creation panel was updated to create, add or delete ribs.

Drawbead and Drawbar Creation Panel

Drawbead and Drawbar creation panel was updated to automatically detect the bead intrusion direction and to provide an option to reverse it.

Resolved Issues

- Holes in components recognized and additional parts in blank nesting.
- Undercheck tolerance has been updated and is now provided as an option in the HyperForm Settings dialog.
- Missing model variables in the HyperForm and HyperStudy incremental interface has been fixed.
- Missing local coordinate system in LS-DYNA Incremental export has been fixed.
- Error in Multi-stage Process trim setup has been fixed.

Back to top

11.0.120 Release Notes - Incremental

Stopper

Stopper has been introduced in an Incremental RADIOSS setup to realistically consider the stoppers that are used to control the motion and the velocity of the binder while it is applying force during forming process.

Tube HydroForming Setup

Tube hydroforming setup has been updated to handle the following use cases:

a) non - axial feeds,

b) starting with dies closed completely, and

c) user defined animation output.

Also, the tube hydroforming setup has been updated to automatically recognize tube, dies and axial feed elements

---

Resolved Issues

- Issues with trim binder when drawbead exists have been resolved.
- Punch and blank holder creation in die module browser fixed to consider drawbeads, drawbars and the state of the binder.
- Mesh is now updated of binder after it is trimmed by the addendum.

Back to top

11.0.120 Release Notes - RADIOSS One Step

Results Initializer

New GUI has been introduced for the batch Results Initializer for ease of use and simplifying the process.

Back to top

11.0.101 Release Notes - Incremental

User Process/Auto Process

New option to define motion and force for each tool in local direction has been added under User Process.

New option added under User Process and Auto Process to set up a forming model using solid elements for the blank, with the exception of trimming process.

New option added under User Process and Auto Process to define thermo-mechanical forming processes, such as hot stamping and elevated temperature forming of aluminum alloys and magnesium alloys.

Thermal option allows setting the tool temperature, initial blank temperature and necessary heat transfer parameters in a user-friendly environment.

Temperature Output in .sta Files

Support of temperature output under .sta file for multi-stage thermal analysis.

Forming Contact TYPE 21

Forming contact TYPE 21 updated to handle better flow of blank material on the tool edge.

Forming contact TYPE 21 updated with damping option to reduce the tool bounce during force control.

Output Animation Files

Option added to output animation files based at user-defined intervals via sensors.

Back to top

11.0.101 Release Notes - RADIOSS One Step

Output Stresses

The option to output stresses in a local coordinate system to enable setting up of springback analysis in a vehicle coordinate system.

Back to top

Highlights of the HyperXtrude 12.0 Release

All HX Profiles

HX Job Manager to launch HX and RADIOSS runs on local machines, remote machines, and E-compute servers.

Metal Extrusion

- Bearing Optimization Interface
- Tool Deflection Analysis Using RADIOSS
- Super Alloy Tube Extrusion Wizard with Glass Lubrication

Polymer Processing

- Viscoelastic Material Model
- Major Improvements to Wizards

Forging

- New solver and interface, HX-Forge, for billet forging
- Forging profile now supports two solvers, Deform and HX-Forge

Contents

12.0 Release Notes - HyperXtrude/All Profiles

HW Desktop

HX Import

HyperXtrude Export

Resolved Issues

12.0 Release Notes - HyperXtrude/Metal Extrusion

Tool Deflection Analysis

Bearing Optimization

Tube Extrusion Wizard

Hensel Spittel Material Model

HX Job Manager

Tool Temperature Data

Glass Properties

Interior Surface BCs

Friction Models

Resolved Issues

12.0 Release Notes - HyperXtrude/Polymer Processing

Meshing Wizards

Plate Meshing Wizard

Coextrusion Wizard

Runner Balancing Wizard

Melt Flow Analysis

Viscoelastic Material Model

Project Browser

Symmetry BC

Interior Surface BCs

Resolved Issues

12.0 Release Notes - HyperXtrude/HX-Forge

HX-Forge Profile

Forging Wizard

Utility Macros

12.0 Release Notes - HyperXtrude / Add Profiles

HW Desktop

HyperXtrude user profiles are now available from HW Desktop. This will enable user to view both the pre-processing (HM) and the results (HV) from the same window.

HX Import

Importing HX data deck using the green arrow button is available and it can be invoked from the standard toolbar of HyperMesh.

HyperXtrude Export

Interface is enhanced to set advanced parameters for using multiple CPUs/cores in a shared memory computer.

Resolved Issues

- If an HM model was imported using feinput, project browser was not recognizing the CAD data. This issue is fixed. Now the browser will accept HM file imported as well as loaded in the standard way.

- Bug fixes to address issues arising from error in the project root, file write permissions, etc. are implemented.

Back to top

12.0 Release Notes - HyperXtrude / Metal Extrusion

Tool Deflection Analysis

This wizard will guide the user to perform tool deformation analysis using RADIOSS solver. It has the following features:

- Step by step guide do the analysis starting with solids
- Support for Pressure/Force loads
- Support for Contact BCs
- Support to do analysis using assumed loads
- Support to do analysis with loads computed by HX solver, by mapping the results on the tool mesh
- Higher order elements are now supported by the wizard. Accuracy of the results computed by RADIOSS solver increase when second order elements are used.

In addition this tool requires no prior knowledge of RADIOSS solver or its native interface to perform the analysis.

Bearing Optimization

Create/Edit BC macro now provides an interface to do bearing optimization with HX solver. This is a direct and faster approach and does not require HyperStudy. This approach groups the bearing control points into design, fixed, and dependent points. By adjusting the design points based on the material flow dynamics, it balances the material flow.

Tube Extrusion Wizard

This wizard will setup model for extrusion of metal tubes using glass coated conical die or a flat die with a glass pad. This setup does not need CAD geometry and it will automatically create the geometry and the mesh.

Hensel Spittel Material Model

User interface for Hensel-Spittel material is added. This will allow to users to create and assign materials with this constitutive model to element components.

HX Job Manager

HX Job Manager is now part of HyperWorks. Until the previous release it was a standalone product that was downloaded separately. HX Job Manager is used launch HX and RADIOSS solver runs. These runs can be launched on local Windows machines, remote Windows/Linux machines, and on PBS servers.

Tool Temperature Data

This data is used to specify the temperature of the tool. Interface will write this data in the GRF file

based on the specified container and die data. If there are other tool components in the model, you have to manually edit the GRF file and modify this data. This is data is relevant only for transient analysis.

Glass Properties

Glass Properties is used to model glass pads used in super alloy extrusion. It is available now from material database window.

Interior Surface BCs

These BCs are used for diagnostic purposes to better understand the material flow in a multi-hole die, as the solver will print the mass and heat flux through these faces. Extrusion Wizard will automatically capture interior faces between two workpiece components having same material and mark them as interior surface boundaries.

Friction Models

BC editor is enabled to support all friction models.

Resolved Issues

- Bugs in determining the bearing starting point have been fixed. Now the macro will ignore the 3D mesh and use only the solids in the model to determine the bearing starting position. In addition, user can modify the detected starting point.
- When importing existing data decks, HX will read the parameter file and update the Extrusion Wizard and Process Parameters cards. This fix includes improvements to bearing profile import.
- Tube Extrusion Wizard:
  - Few meshing related bug fixes were implemented. Tolerances used are made unit dependent to resolve meshing issues. In addition, additional checks are implemented to verify the dependency in meshing parameters.
  - For rod extrusion, there is no mandrel and hence no mandrel BCs. Application error related to this is issue is fixed
  - Data checks to validate the geometry data are implemented
  - Separated Mesh and BC creation process steps in Tube Extrusion Wizard for better control
  - Redesigned Model/Process information GUI in Tube Extrusion Wizard
- BC check for undefined boundary faces assumed that for every load collector there will be a corresponding collector with BC faces and the vice versa. This was throwing an error if user had accidentally deleted one of them. This is issue is fixed. In addition, when the new BCs for undefined faces are created – they are created with correct conditions and the user will be not be able export a model that contains BCs created by "check undefined BC". User will have to inspect and rename this BC to export the model.
- Checks to ensure billet length specified is greater than butt length is not relevant to steady state analysis. This data check is streamlined now and the underlying errors in the default values (unit dependency) are corrected.
- Bug fixes to control point selection panel. Even if user selects points that are not part of the bearing profile, they will be automatically rejected.
- Interface will support only the standard indirect extrusion, where the dummy block and container move along the extrusion axis. Indirect extrusion with moving die will be supported in the next release.

- Symmetry faces on tool elements are now organized into separate components to overcome errors in detecting undefined boundary faces.

Back to top

12.0 Release Notes - HyperXtrude / Polymer Processing

Meshing Wizards

All the model setup wizards in "Polymer Processing" are enhanced to efficiently select the components and boundary faces. In addition, all wizards derive their specific versions form this common framework and hence, provides a similar look and feel.

Plate Meshing Wizard

Upgrades to this wizard include:

- Computation of exit velocity and extrusion ratio
- Starting the wizard from an existing mesh
- Support for including the tool in the model

Coextrusion Wizard

A new wizard is implemented to guide user systematically starting from a CAD model to setup coextrusion models with inserts.

- It will compute and show the exit velocity and extrusion ratio for each material layer.
- It supports starting the model from an existing mesh
- It supports meshing and BC creation for Tool

Runner Balancing Wizard

The Runner Balancing Wizard implemented in this release is used to create mesh for injection molding runner balancing analysis. This tool creates high quality meshes and can handle complex runner systems.

Melt Flow Analysis

A wizard to guide the model setup of polymer melt flow (or a non-Newtonian fluid) in any three dimensional conduit is implemented.

Viscoelastic Material Model

Material Database now supports Criminale—Ericksen—Filbey viscoelastic material model.

Project Browser

Enabled Project Browser to manage projects and runs for Plate Meshing, Coextrusion and Melt Flow Analysis wizards.

Symmetry BC

Polymer meshing wizards now provide an option to create symmetry boundary conditions. It is supported even for models that start from an existing mesh.

Interior Surface BCs

Plate Meshing, Coextrusion and Melt Flow Analysis wizards captures Interior Surface BCs between two Polymer components having same material.

Resolved Issues

- Fixes to unit conversion errors when specifying melt flow rate in non-standard units (such as, mm/min, cm/min, m/min, etc.).
- When the model has insert, interface will automatically write the tool velocity data packet. This data is expected by the solver.
- Exits and FreeSurface BCs are organized based on XY values in Plate Meshing, Coextrusion and Melt Flow Analysis Wizards.

Back to top

12.0 Release Notes - Forging / HX-Forge

HX-Forge Profile

The Forging user profile now supports two solvers: Deform and HX-Forge. Latter is a new solver targeted for billet forging applications.

Forging Wizard

A new wizard is implemented to guide user step by step to create model data. In this release, this wizard will create model for “round-to-round” and “square-to-square” billet forging analysis.

Utility Macros

New utility macros are added for creating tool sets and process builder in order to set up forging model. With these tools, user can manually create and setup models for analysis.

Back to top

Highlights of the HyperMold 12.0 Release

General

- Improvements to Meshing Wizards
- Menu Panels Redesigned to Include All Essential Buttons

MoldFlow User Profile

- Fusion Mesh Ratio
- Flow3D Wizard
- Feed System Wizard
- Solid Meshing Wizard
- Create Feed System Enhancements
- Extract Centerline from Feed Solids
- Macro for Property Card Assignment
- Resolved Issues

MoldEx3D Solid User Profile

- MFE Import
- Solid Meshing Wizard
- Resolved Issues

MoldEx3D Shell User Profile

- A New Sub-profile
- Thickness Extraction
- Runner Design
- Extract Centerline from Feed Solids
- Create Feed System Enhancements
- Cooling Circuit
- MSH Import
- Symmetry BC
- Resolved Issues

RTM User Profile

Data Import

Resolved Issues

Contents

11.0.130 Release Notes

General Updates

MoldFlow User Profile

Moldex3D Solid Interface

Moldex3D Shell Interface

11.0.120 Release Notes

Resolved Issues

11.0.101 Release Notes

MoldFlow

12.0 Release Notes - General

Menu Panels

Menu panels of sub-profiles are redesigned to include all the essential panel buttons.

Edge Based Mesh Density Control

A powerful automatic 2D meshing technology to mesh a set of surfaces has been implemented. This method automatically adjusts the density of the created 2D mesh on all edges based on a set of built in automation rules. This edge based surface mesh algorithm improves the mesh quality of the surfaces. This algorithm is implemented in all HyperMold automatic surface mesh macros like Flow3D, Moldex3D Wizards, etc.

Back to top

12.0 Release Notes - MoldFlow

Fusion Mesh Ratio

Mesh match ratio of fusion models using a new algorithm is now computed using a C++ DLL. This new algorithm is more accurate and agrees well (within 10%) of Moldflow's estimate. In addition, it is very fast – it can estimate for quarter million element model in less than a minute, which is more than 100 times faster than old approach.

Flow3D Wizard

Enhancements to improve meshing accuracy.

Built-in access to Feed system wizard

Feed System Wizard

To design feed system in minutes using a simple step by step approach.

Feed system is now rendered using solids instead of surfaces and also faster than before. This will help the user to transition to a 3D runner system from a 1D system.

Solid Meshing Wizard

This wizard is used to generate a 3D mesh in a selected set of solids in easy steps. User can select the meshing method for each solid: solid map, tetra mesh or CFD boundary layer mesh. In addition, the wizard automatically computes the default element size and the shared surfaces between the solids can be meshed without a boundary layer.

Create Feed System Enhancements

Feed system macro is enabled to create solids for curved feed lines.

Extract Centerline from Feed Solids

A new macro to extract the centerline from feed system solids has been implemented. This axis can be used as a feed line in feed system creation.

Macro for Property Card Assignment

This macro assigns the existing Moldflow property card to the selected elements.

Resolved Issues

- **Assign Thickness:** Thickness extraction was not working correctly when multiple surfaces were selected.
- **Feed System:** Error in creating tapered feed lines is fixed.
- **Feed System Wizard:** Enhanced to automatically proceed to the next step in the wizard.
- **Flow3D Mesh Wizard:** Enhanced to handle the latest CFD mesh options.
- **CFD Mesh:** Naming the collector containing the CFD Mesh collector name is now derived from the original component name and the solid ID.
- **Occurrence Number:** Error in plotting the occurrence number plotting is fixed.

Back to top

12.0 Release Notes - Moldex3D Solid

MFE Import

Import is made more comprehensive by including all the relevant property cards and boundary conditions. Import is also now speeded-up by removing the bottlenecks in the process.

Solid Meshing Wizard

This wizard is used to generate a 3D mesh in a selected set of solids in easy steps. User can select the meshing method for each solid: solid map, tetra mesh or CFD boundary layer mesh. In addition, the wizard automatically computes the default element size and the shared surfaces between the solids can be meshed without a boundary layer.

Resolved Issues

- **Feed/Coolant Channel:** Fixed the application error occurred while finding element connectivity for Feed/Coolant channel elements.

Back to top

12.0 Release Notes - Moldex3D Shell

A New Sub-profile

This sub-profile comes with macros to create, setup, an export models for Moldex3D/Shell.

Thickness Extraction

Users can extract thickness from the midsurface and automatically create property cards for cavity elements.

Runner Design

Macros to create and setup runner design are available in the interface. This includes surface rendering of runner geometry for visualization. Runner system is created on lines with 1-D beam elements.

Extract Centerline from Feed Solids

A new macro to extract the centerline from feed system solids has been implemented. This axis can be used as a feed line in feed system creation.

Create Feed System Enhancements

Feed system macro is enabled to create solids for curved feed lines.

Cooling Circuit

Macros to create and setup cooling channels are available in the interface. This includes surface rendering of channel geometry for visualization. Cooling channels are created on lines with 1-D beam elements.

MSH Import

Existing Moldex3D Shell data decks can be imported into the user profile.

Symmetry BC

Support to specify symmetry nodes has been implemented.

Resolved Issues

- **Export:** Fixed exporting issues from Midplane Wizard.

Back to top

12.0 Release Notes - RTM

Data Import

Existing RTM data decks can be imported back into HyperMesh. This release provides a limited support for these imports and the main goal is to import the 3D elements and the boundary faces.

Resolved Issues

- **Material Database:** Added the missing material parameters for perform: density and compressibility.
- **Parameter:** Automatically writing “ReservedMemorySize”, which is a required data.

Back to top

11.0.130 Release Notes

General Updates

- Menu panels of sub-profiles are redesigned to include all the essential panel buttons

---

MoldFlow User Profile

- Error in creating tapered feed lines is fixed.

Moldex3D Solid Interface

- Runner Meshing Wizard is enhanced to include CFD tetmeshing. Based on the user selections, it will mesh the different regions using solid map, volume tetmeshing and CFD tetmeshing.
- MFE Import is made more comprehensive by including all the relevant property cards and boundary conditions. Import is also now faster by removing the bottlenecks in the process.

Moldex3D Shell Interface

- **MSH Import:** Existing Moldex3D Shell data decks can now be imported into the user profile.

Back to top

11.0.120 Release Notes

Resolved Issues

- **Assign Thickness:** Thickness extraction was not working correctly when multiple surfaces were selected. This issue is fixed.

Back to top

11.0.101 Release Notes

MoldFlow

A new Midplane Wizard to create midplane models has been added. This wizard can start with models that have volumes (solids/surfaces), midplane surfaces, or midplane mesh. When starting from a volume, it can extract midsurface from the volume and map thickness on the midsurface to create property cards.

Improvement to the 2D meshing algorithm used by all the wizards to control the edge densities has been added. Users can specify the minimum and maximum element size and the wizard will scan through all the surface edges, sort them by length and interpolate the edge density based on a set of rules.

Back to top

Engineering Solutions 12.0 Release Notes

HyperWorks Engineering Solutions is a customized and highly automated environment for modeling and visualization environment of NVH, CFD, Crash, Durability, Suspension and Drop Test applications.

CFD	High quality tools for CFD applications enabling the engineer to perform modeling, optimization and post-processing tasks efficiently.
NVH	HyperWorks environment customized for automotive full vehicle NVH modeling and analysis needs.
Crash	A tailored environment in HyperWorks that efficiently steers the Crash CAE specialist in CAE model building, starting from CAD geometry and finishing with a runnable solver deck in both solvers RADIOSS (Block Format) and LS-DYNA.
Drop Test	An automated solution that allows you to either simulate a single drop test or a choice of multiple iterations with the aim of finding the sensitivity of process variables like initial orientation and drop height in a typical drop test by controlling the run parameters and conditions with ease.
Durability Director	Solver-neutral, process-oriented customization of HyperWorks that addresses many of the challenges associated with assessing the fatigue life of mechanical components.
Suspension Director	Industry specific solution that is integrated with MotionView and utilizes many aspects of HyperWorks to assist with the engineering of vehicle suspensions.
HyperCrash	CAE pre-processor tool developed to support the non-linear finite element solver, Altair RADIOSS.
AcuConsole	Powerful, yet easy to use GUI-Based Computational Fluid Dynamics (CFD) pre-processor for AcuSolve.

Highlights of the CFD 12.0 Release

- Auto Collision Detecting for Boundary Layer Growth
- Advanced Parameters for Boundary Layer Generation

Contents

11.0.130 Release Notes

Option to Specify Fluid and Non-fluid Volumes

Simple Boundary Selection

2D BL Meshing (Smooth)

11.0.120 Release Notes

Nastran Export for PowerFlow, Nastran (PowerFlow)

11.0.101 Release Notes

Periodic Surface Meshing

Tetra Mesh Optimization

New CFD Surface Mesh Criteria

Automatic BC Shell Update for CFD Tetra Meshing

AcuSolve Case Setup

Shell Meshing with Boundary Layers

Automatic Boundary Layer Reduction

Multi Threading for Boundary Layer Generation

Hex-core Meshing Utility

CGNS Format Import

CFD User Profile

Mesh Statistics for Mesh Export

Particle Tracking (Post Processing)

Support of Particles in the Ensight Reader (Post Processing)

Moving Mesh Support (Post Processing)

New Ensight Reader Option (Post Processing)

12.0 Release Notes

Auto Collision Detection for Boundary Layer Growth

The boundary layer (BL) reduction step is now included into the BL generation routine. During boundary layer generation a collision check is performed and the boundary layer thickness is adjusted dynamically.

Advanced Parameters for Boundary Layer Generation

Three new options to control the boundary layer growth mechanism have been introduced, including a quality focused and a meshing time focused set of predefined parameters. Selecting the user defined option will give access to several boundary layer parameters which can be adjusted individually for each application.

Back to top

11.0.130 Release Notes

Option to Specify Fluid and Non-fluid Volumes

Option to specify fluid and non-fluid volumes: For the fluid volumes, a tetra mesh including a boundary layer will be generated; whereas the non-fluid domains will be tetrameshed only.

Simple Boundary Selection

An option to simplify the user input for CFD tetrameshing has been added; for example, cases containing internal control surfaces.

2D BL Meshing (Smooth)

The utility has been enhanced by the following features:

- Selector for "w/o BL (fixed)" and "w/o BL (remesh)": Non BL boundary regions can be defined as "fixed" or "remesh". This is relevant if two surfaces are meshed sequentially and a matching interface is required.
- Elements to region component: Option to place the newly generated surface elements into the region's component or into two new components (old behavior) has been added.
- Boundary layer reduction: Automatic and manual reduction of the boundary layer thickness to avoid intersection has been included. This is similar to the mechanism available in the CFD Tetramesh panel.

Nearly Unoffsetable Nodes

The boundary layer generation for complex geometry (nearly unoffsetable nodes) has been improved drastically. A user defined node collapse angle, defining the threshold for boundary layer collapse, has been introduced.

Surface Remeshing for Refinement Box

Option to remesh surface elements if they are included in a refinement box has been improved. The

surface remesh will be performed with the elements size assigned to the refinement box.

Back to top

11.0.120 Release Notes

Nastran Export for PowerFlow, Nastran (PowerFlow)

The Nastran export format from HyperMesh for surface shell elements has been enhanced. The component names and element grouping will be maintained during import into PowerFlow.

Back to top

11.0.101 Release Notes - CFD

Periodic Surface Meshing

A utility to generate rotational and/or translational periodic surface meshes has been included.

Tetra Mesh Optimization

The “Tetra Mesh Optimization” tool will improve the mesh quality of an existing tetra mesh with respect to user selected element criteria and thresholds. Several criteria can be selected at the same time. Surface remeshing and refinement boxes can be considered as well during the optimization.

New CFD Surface Mesh Criteria

The concept of the “vol skew” element criteria has been applied to tria surface elements to define the “area skew” criteria.

Automatic BC Shell Update for CFD Tetra Meshing

In the CFD Tetramesh panel, the user has the option to define some boundary regions as “float”, allowing HyperMesh to swap edges of shell elements during the meshing process. Activating the new option “post-update input shells” will automatically update the initial surface mesh to match the volume mesh.

AcuSolve Case Setup

A utility to setup an AcuSolve input deck has been included. Currently only a limited number of input parameters are supported.

Shell Meshing with Boundary Layers

Both meshing algorithms for “2D Native BL (planar)” and “2D Smooth BL” have been further enhanced to ensure a good quality of the resulting mesh.

Automatic Boundary Layer Reduction

The utility to perform the automatic boundary layer thickness reduction (also known as “generate BL

thickness”) has been improved. The effect of normal smoothing has been included in the proximity check.

Multi Threading for Boundary Layer Generation

The algorithm for boundary layer generation has been improved by using multi-threading technology, resulting in a decrease of meshing time on multi core computers.

Hex-core Meshing Utility

The option to reduce the boundary layer thickness manually or automatically has been added to the hex-core meshing utility for Cartesian as well as curvilinear grid types.

CGNS Format Import

The CGNS file format import has been enhanced. During import of a CGNS mesh file, the boundary region tagging will be maintained.

CFD User Profile

Added more functionalities from the standard HyperMesh user profile to the CFD user profile. A button to launch the “AcuSolve Job Launcher” dialog has also been added to the CFD toolbar.

Mesh Statistics for Mesh Export

For the “mesh to file” functionality in the CFD Tetramesh panel, the export of a mesh statistic ASCII file has been added. The file contains the element quality distribution for several mesh criteria.

Particle Tracking (Post Processing)

Streamlines can now be animated with particles traveling along the streamlines. Various contour and animation options for the particles have been implemented.

Support of Particles in the Ensight Reader (Post Processing)

Particle data (e.g. modeling of dust particles in a flow) existent in an Ensight data set can be animated. The particles are visualized as spheres and can be contoured with respect to particle diameter.

Moving Mesh Support (Post Processing)

The Ensight result reader has been enhanced to support transient data including mesh deformation (e.g. sliding mesh, ALE).

New Ensight Reader Option (Post Processing)

Using the option “auto” for the Ensight reader, Ensight result files generated by SC/Tetra and other CFD solvers (e.g. AcuSolve, Fluent) can be imported.

Back to top

Highlights of the NVH 12.0 Release

New Assembly Network View (PSO only)

New Job Manager (PSO only)

New Connection Property State Mapping Capability (PSO only)

Enhanced Analysis Manager (PSO only)

Enhanced Tapoint Mapping Capability (PSO only)

Enhanced ID Manager (PSO only)

Enhanced Unit Input and Random Frequency Response Process Managers

Enhanced General Frequency Response Process Manager

New Energy Distribution Utility

Enhanced Transfer Path Analysis Utility

Enhanced Modal/panel Participation Utility

Enhanced Design Sensitivity Analysis Utility

Contents

11.0.130 Release Notes

- Linux 64bit Support
- ID Manager
- Tagpoint Creation Tool
- Enhanced Assembly Browser Context Menu
- Enhanced Prepare Module Mode
- Integrated Diagnostics Tool
- DSA Utility
- Enhanced Transfer Path Analysis Utility
- Enhanced Model/Panel Participation Utility

11.0.120 Release Notes

- Isolate Checkbox in Module Manager
- DOF Information
- Solver Deck Review
- Prepare Module Mode Enhancements

11.0.101 Release Notes

- Loadcase Setup Process Manager Templates

Introduction

Starting from 12.0, NVH Director features will be reorganized into two packages, one includes general NVH features which will continue to be available as a part of the standard HyperWorks, another includes high productivity NVH features which will be available through the Packaged Solution Offerings (PSO) NVH Director product, and will not appear in the standard HyperWorks without a PSO license.

1. Standard HyperWorks Package
   - Meshing
     - Cavity Mesher
     - Coarse Mesher
   - Loadcase setup process managers in generic HyperMesh mode
   - All RADIOSS solver NVH functionalities
   - Post-processing utilities (all but 'Integrated Diagnostics')
     - Modal/panel Participation
     - Grid Participation
     - Energy Distribution
     - Transfer Path Analysis
     - Design Sensitivity Analysis
     - Order Analysis
2. Packaged Solution Offerings (PSO) Package
   - HyperMesh - NVH user profile under 'Engineering Solutions'
     - Assembly Browser
     - Network View
     - Analysis Manager
     - Job manager
     - Loadcase Setup Process Managers in NVH User Profile Mode
   - HyperView/HyperGraph - Integrated Diagnostics utility

Back to top

12.0 Release Notes

New Assembly Network View (PSO only)

- An alternative way to work with modules and connections in assembly
- Best for reviewing assembly topology and connectivity

New Job Manager (PSO only)

- Provide a list of all jobs created as a part of a project

- Provide a record of why each job was run and how the model was setup
- Provides easily access to result file generated in each job

---

New Connection Property State Mapping Capability (PSO only)

- Connector property state linked to entries in CSV file thru import
- Common rate definitions for multiple connections
- Connection CSV import/export capability enabling automatic connection update and re-creation

---

Enhanced Analysis Manager (PSO only)

- Capability to create jobs and associate package of files for traceability and post-processing purposes
- Capability to submit jobs to user specified solvers

Enhanced Tagpoint Mapping Capability (PSO only)

- Identify nodes from different modules based on proximity as candidate for module and connection creation
- Import and export tagpoints and connections using CSV files

---

Enhanced ID Manager (PSO only)

- Export to module ID CSV includes module hierarchy information
- Automated module definition update and re-creation through import of module ID CSV

Enhanced Unit Input and Random Frequency Response Process Managers

- Support for user defined subcases involving multiple inputs

---

Enhanced General Frequency Response Process Manager

- Support for multiple user defined subcases
- Support for diagnostic output at ofreq frequencies

New Energy Distribution Utility

- Visualize vehicle system energy distribution by proper comparator groups in the assembly
- Capability to create comparator groups using assembly information from job files created using the Analysis Manager
- Supports energy output for finite elements, as well as system and component modes

---

Enhanced Transfer Path Analysis Utility

- Support for the new RADIOSS AutoTPA (one-step TPA) output in H3D file

- Support for multiple control volumes
- Support for multiple subcases
- Support for Spindle Loads (simultaneous multiple principal vector subcases)
- Support for Engine Loads (simultaneous multiple RPM subcases)
- Support for assumed forces

Enhanced Modal/panel Participation Utility

- Support for the new RADIOSS component modal participation output in H3D file
- Support for associating a normal modes animation file as diagnostic results

---

Enhanced Design Sensitivity Analysis Utility

- Support for ranking design variables by ‘Magnitude sensitivity’
- Enhanced design variable labels using assembly information
- Support for output generated using the new RADIOSS DSA request in H3D file

Back to top

11.0.130 Release Notes

Linux 64bit Support

Added Linux 64-bit support for all pre and post processing capabilities.

ID Manager

Added ID Manager which provides a numerically sort-able table of IDs assigned to each module. It can be invoked either from the ID tab of the Module Manager dialog, or from the ID view of the Assembly Browser.

Tagpoint Creation Tool

Added Tagpoint Creation tool which allows searching for candidate points among nodes between module pairs, based on proximity, to be used as tagpoints. This tool is added to the NVH toolbar next to the LCS Manager.

Enhanced Assembly Browser Context Menu

- Moved two Import XML sub-menu items, XML Full and XML Only, to two corresponding root menu options: Import XML and Display and Import XML only.
- Moved four Manage Module sub-menu items, Representation, Tagpoints, ID, and States to four corresponding root menu options: Manage Representations, Manage Tagpoints, Manage ID, and Manage States.

Enhanced Prepare Module Mode

Added Skip renumbering checkbox in the Module ID summary dialog, which allows user to keep existing module entity IDs as they are while preparing the module.

Integrated Diagnostics Tool

Added Integrated Diagnostics post-processing utility which introduces the capability to link diagnostic results to relevant the responses so that user can perform diagnostic analysis on the response using dynamic right clicking context menu to visualize available diagnostic results.

DSA Utility

Added DSA post-processing utility which allows visualization of design sensitivity data and studying effect of design variables on responses using linear projection.

Enhanced Transfer Path Analysis Utility

- Added support for RADIOSS Auto TPA output (using the 'PFPATH' output request card) in a single H3D file.
- Added support for cases where no force file is provided. Assumed constant forces will be generated internally by the utility to allow a special TPA analysis to proceed.

Enhanced Model/Panel Participation Utility

Added support for component modal participation output from RADIOSS to H3D file.

Back to top

11.0.120 Release Notes

Isolate Checkbox in Module Manager

Added Isolate checkbox next to module selection navigation up/down arrows in Module Manager. When checked, the selected module is isolated in the 3D graphics window. The same functionality is also added to the Connection Manager.

DOF Information

Added degree-of-freedom information to the property type column in the property view of the Connector Browser.

Solver Deck Review

Added **solver deck review** button in lumped parameter representation definition tab of the module manager and the template loadcase definition dialog in the Analysis Manager.

Prepare Module Mode Enhancements

- Auto load display representation of the last module prepared prior to exiting the prepare module mode.

- Added Add PLOTEL subtab to the prepare module tab which helps user with PLOTEL display elements.
- The Module ID summary dialog now returns after user executes an ID management action to show the modified IDs in an abbreviated form. The same abbreviated dialog will now also appear when entering the prepare module mode, if the ID range of the module had already been validated, in which cases, the ID summary dialog was previously skipped and not shown.
- Added a quick navigation button to the bottom of the prepare module tab to quickly open the tagpoint tab of the module to add or delete tagpoints.

Back to top

11.0.101 Release Notes

Loadcase Setup Process Manager Templates

Five loadcase setup process manager templates are added to the RADIOSS (Bulk) solver profile:

- Normal modes
- CMS SE Generation
- Unit input frequency response
- Random frequency response
- General frequency response

Four loadcase setup process manager templates are added to the Nastran solver profile:

- Normal modes
- Unit input frequency response
- Random frequency response
- General frequency response

Back to top

Highlights of the Crash 12.0 Release

- IP Head Impact Process Manager for RADIOSS

Contents

11.0.130 Release Notes

Team Center - HyperMesh Assembly Browser

IP Head Impact Process Manager for LS-DYNA

Results Initializer

11.0.120 Release Notes

Updated Graphical User Interface

11.0.101 Release Notes

TeamCenter PLMXML Files

Creation of Self-contained Include Files in Ls-Dyna User Profile

Finding Cross References

Autopitch

Crash Post-processing: IIHS

12.0 Release Notes

IP Head Impact Process Manager for RADIOSS

The IP Head Impact Process Manager is a tool that generates a deck for the simulation of the head impact on an instrument panel. It conforms to the interior safety regulations FMVSS 201 and ECE R 21. IP Head Impact automates the process with minimal input from the user, therefore reducing the deck generation lead time and enhancing repeatability.

- Specify seating reference point for positioning
- Specify vehicle motion in positive or negative X direction
- Calculation of standard hit point zones as per FMVSS 201 and ECE R21
- Display this zone showing only the elements in the zone or highlighting them

- Ability to scale this zone
- SAE J921 conformant angle calculation procedure
- Additional angle calculation method-normal at the point of impact
- Be able to select hit points manually or by importing an existing .csv file generated by the same tool
- Review hit points with or without penetration removal
- Export solver decks for all or selected hit points

Back to top

11.0.130 Release Notes

Team Center - HyperMesh Assembly Browser

- PLMXML assembly files from TeamCenter can now be directly read in HyperMesh.
- This is supported in the Engineering Solutions user profile of RADIOSS and LS-DYNA.
- TeamCenter Configuration is required; please consult your TeamCenter support for setting up the configuration for PLMXML export.
- Geometry files in the assembly tree can be directly batchmeshed using the multi-threading capabilities of the Batchmesher.
- Allows for various representations for each module in the assembly tree.
- Batchmeshed parts can be exported at module level as solver files and or HyperMesh files.
- Connections in mcf format can be directly imported and realized.

IP Head Impact Process Manager for LS-DYNA

The IP Head Impact is a tool that generates a deck for the simulation of the head impact on an instrument panel. It conforms to the interior safety regulations FMVSS 201 and ECE R 21. IP Head Impact automates the process with minimal input from you, therefore reducing the deck generation lead time and enhancing repeatability.

Results Initializer

Results Initializer has been updated to support Tailor welded Blanks.

Back to top

11.0.120 Release Notes

Updated Graphical User Interface

New GUI has been introduced for the batch Results Initializer for ease of use and simplifying the process.

Back to top

11.0.101 Release Notes

TeamCenter PLMXML Files

TeamCenter PLMXML files are now supported in HyperMesh for batchmeshing and assembly:

- Multi-core batchmeshing is seamlessly exposed through the Assembly Browser
- Ability to initiate batchmeshing by just loading the assembly file (PLMXML) without loading the CAD parts in HyperMesh
- Metadata related to material, thickness and component ID's are automatically assigned in HyperMesh
- Param and Criteria files are assigned based on metadata in the PLMXML file
- Delta PLMXML export is supported that can be imported in TeamCenter for Simulation

TeamCenter PLMXML file import options are supported in the Assembly Browser. The Assembly Browser supports the following tasks:

- Initiating batchmeshing by specifying number of processors on multi-core machines
- Assign/Change/Edit param and criteria file for meshing
- Save individual component HM files after batchmeshing
- Change from CAD to mesh representations easily

Creation of Self-contained Include Files in Ls-Dyna User Profile

In the past when moving comps to across includes, we provided options to move nodes and elements. Now there is an option to create a self-contained include file with no external references. The option to not move material and properties associated with the components is also available.

Finding Cross References

A powerful feature to find cross references across includes for Dyna and RADIOSS has been introduced. It allows you to understand the dependency in Include files.

The right-click context menu of an Include file in the Model Browser displays the option "Include xref." This shows the references with external Include files.

Autopitch

AutoPitch tool to determine weld locations based on flanges is provided under the Connectors menu for spot welds.

Crash Post-processing: IIHS

In the Vehicle Safety Tools preferences, there are now two new templates to plot IIHS front intrusion and IIHS side structure intrusion plots for RADIOSS and Dyna.

IIHS Front Intrusion works on:

- T01 file of RADIOSS

- Nodout, binout and d3plot files of Dyna
- HyperView H3D files

IIHS Side Intrusion works on:

- A0 files of RADIOSS
- d3plot files of Dyna
- HyperView H3D files

Back to top

CAE Result Player

HyperView Player	Plug-in and stand-alone utility to share and visualize 3-D CAE models and results

CAD and Solver Interfaces 12.0 Release Notes

Use the links below to view the HyperWorks 12.0 release notes for the CAD and solver interfaces.

RADIOSS (Bulk Data Format) and OptiStruct

RADIOSS (Block Format)

Abaqus

ANSYS

LS-DYNA

Nastran

PAM-CRASH 2G

Highlights of the Abaqus Solver Interface 12.0 Release

- Abaqus 6.12 Reader
- Results on Element Faces
- Random Response Analysis Enhancements

Contents

11.0.130 Release Notes

Support of Generated Nodes and Elements

CONTACT CONTROLS ASSIGNMENT

New Parameters for *CONNECTOR HARDENING

DYNAMIC Analysis Procedure

New Parameters for *FILTER (Explicit Interface)

*PREPRINT, CONTACT

Resolved Issues

11.0.120 Release Notes

Virtual Crack Closure Technique

Improved Displacement System Support

Ply and Laminate Support

Composites Panel

Systems Panel

Resolved Issues

11.0.101 Release Notes

New Element Types

New Thermal Loads

Analysis Procedures Enhancement

*CAVITY DEFINITION

*SURFACE PROPERTY

*SURFACE INTERACTION

Name Change

Update Orientation/Offset Tool

Contact Manager Enhancements

*MONITOR

Sorting Algorithm for Auto Contact Results
Midsurface Thickness Map Utility
*REBAR LAYER
Resolved Issues
Abaqus 6.10-EF and 6.11 Readers
Sorting Algorithm for Auto Contact Results

12.0 Release Notes

Abaqus 6.12 Reader

The Abaqus 6.12 reader has been added for Windows 32/64bit and Linux 64bit. HyperWorks 12.0 also supports these versions: 6.10, 6.10-EF, 6.11.

Results on Element Faces

A new reader option called **Support element face results** has been added to the ODB reader options. If set to **true**, for each *SURFACE definition a new component will be created. The component name matches the name of the surface definition. Post-processing of results which are available on element faces is only possible this way. The new option is also helpful for post-processing of contact results, as surfaces can be shown isolated from the rest of the model.

Random Response Analysis Enhancements

For random response analysis, the power spectral densities (PSD) and root mean square (RMS) of von Mises stresses are calculated based on a publication of Daniel J. Segalman, et al. (1998). The calculation time can be high for big models. In this case, results can be extracted in batch mode using HVTrans.

Back to top

11.0.130 Release Notes - HyperMesh

Support of Generated Nodes and Elements

Support of generated nodes and elements has been added (*NGEN, *NFILL and *ELGEN). Support for node generation is limited to straight lines and rectangular systems. NFILL is supported in its basic form to create equally distributed nodes between node sets.

CONTACT CONTROLS ASSIGNMENT

New types EDGE TRACKING and ENHANCED EDGE TRACKING have been added for explicit interface. The parameter CONTACT THICKNESS REDUCTION is available with this release as well.

New Parameters for *CONNECTOR HARDENING

New parameters for *CONNECTOR HARDENING:

- EXTRAPOLATION
- REGULARIZE
- RTOL
- RATE FILTER FACTOR
- RATE INTERPOLATION

Except the first parameter all others are available in explicit template only.

DYNAMIC Analysis Procedure

The DYNAMIC analysis procedure got extended by the APPLICATION parameter in standard interface. It can be set to TRANSIENT FIDELITY, MODERATE DISSIPATION and QUASI-STATIC.

New Parameters for *FILTER (Explicit Interface)

New parameters have been added to the *FILTER keyword (explicit interface):

- OPERATOR
- LIMIT
- HALT
- INVARIANT

---

*PREPRINT, CONTACT

DEBUG has been added as setting to *PREPRINT, CONTACT. This feature is not documented in Abaqus documentation, but has been requested by users.

Resolved Issues

- Handling of ZRECTANGULAR systems has been revised. Node and coordinate based systems are always converted to RECTANGULAR systems on import. The conversion did not work correctly in some cases. Upon creation of a node based system we write a ZRECTANGULAR card in case a user defines the system as required in the manual (x-, z-, xz- axis combination in the Systems panel).

Back to top

11.0.120 Release Notes - HyperMesh

Virtual Crack Closure Technique

A set of keywords to support Virtual Crack Closure Technique (VCCT) has been added.

Standard interfaces:

- INITIAL CONDITIONS, TYPE=CONTACT (interfaces)
- DEBOND (group)

- FRACTURE CRITERION (group; part of DEBOND card)
- CONTROLS, TYPE=VCCT LINEAR SCALING (group)

Explicit interface:

- COHESIVE BEHAVIOR (group; part of SURFACE INTERACTION card)
- FRACTURE CRITERION (same as the above)

Improved Displacement System Support

The system card image shows now, to which node set it is assigned.

You can now unassign a node set, by either removing the TRANSFORM flag from the card image or by assigning it back to the global system. This can be done from the System panel by selecting the global system in the lower left side of the GUI.

Ply and Laminate Support

Both entities are now enabled for the Abaqus interface. More details are provided in the generic HyperMesh section of these release notes and in the plies and laminate documentation of the HyperMesh User Guide.

3D ply visualization including ply orientation is now supported for shells and continuum shells.

Laminate definitions can now be realized into Abaqus properties. Pre-condition is that a composite property has to be assigned to the elements. Both, shell and shell general section definitions have been enhanced to interact with ply and laminate entities.

Composites Panel

The normals subpanel in the Composites panel has been enhanced to support continuum shells (currently only available in Abaqus interface).

- In addition to regular shell elements ‘display normals’ can now show the default stacking direction of continuum shells based on their numbering scheme. This button does not consider the STACK DIRECTION setting on the element property.
- ‘Display stack direction’ accounts for the STACK DIRECTION parameter setting of the element property for review. Supported are the settings in all three isoparametric directions (1, 2, 3) but also the system reference).
- Review options are the same as with regular shells: vector and color display.
- Two options to adjust the default stacking direction for continuum shells are available now: Selected elements can either be adjusted by selecting a reference element or a user can specify the base face by two diagonal or three base nodes.

Systems Panel

The assign subpanel of the Systems panel has a new option called attach nodes as set for the Abaqus interface. Now a user can simultaneously assign nodes to a displacement system and simultaneously create a node set.

Resolved Issues

- HM comments for *ORIENTATION have not been honored upon import when the system definition is based on nodes or is defined without the origin.

- The import reader now accounts for reference systems assigned to the nodes of a node based *ORIENTATION definition. Previously these systems have been shown in a wrong position, but no information has been lost.
- Rigid body property card: ‘None’ as setting of this card image has been lost in 11.0.101 release. This setting is used to create a rigid body card which is not pointing to an element set or a component, but to a tie or pinned node set, for example. This has been fixed.

Back to top

11.0.101 Release Notes - HyperMesh

New Element Types

- S3RT.
- S4RT
- SC6RT
- SC8RT

New Thermal Loads

- *CRADIATE
- *RADIATE
- *SRADIATE

Analysis Procedures Enhancement

Analysis procedures *COUPLED TEMPERATURE-DISPLACEMENT and *DYNAMIC TEMPERATURE DISPLACEMENT can now be set in the load step card image or from the Step Manager.

***CAVITY DEFINITION**

Abaqus interface now allows the definition of a cavity definition problem: *CAVITY DEFINITION is a new interaction accessible through the Interactions panel. The surface interaction module of the Contact Manager has also been enhanced to define the cavities. *RADIATION VIEWFACTOR calculation parameters can be set in the load step card image.

***SURFACE PROPERTY**

*SURFACE PROPERTY is a new property in the Standard templates. This includes the dependent keyword *EMISSIVITY. Also, the *SURFACE card has been enhanced to refer to the new property.

***SURFACE INTERACTION**

*GAP CONDUCTANCE, *GAP HEAT GENERATION and *GAP RADIATION keywords have been added to the *SURFACE INTERACTION card. They can be defined either in HyperMesh or in the Contact Manager (Surface Interaction tab).

Name Change

The property type “Contact properties” has been renamed into “Interface properties.”

Update Orientation/Offset Tool

On the Utility Menu, the new tool ‘Update orientation/offset’ has been introduced. It reads the orientation and offset definition from all beam property cards and updates the element database. That way the 3D visualization and model definition can be synchronized and will exactly display what is getting exported.

Contact Manager Enhancements

The GUI for pretension section definition in the Contact Manager has been significantly enhanced. An automatic functionality now creates the pretension node for a definition by 1D elements. If the bolt is modeled by solids it can additionally create the surface cutting through the bolt section.

While creating the pretension definition, you can directly apply assembly loads in the same GUI.

---

*MONITOR

*MONITOR can now point to a node set, too. The template as well as the Step Manager has been updated to support this

---

Sorting Algorithm for Auto Contact Results

In 10.0 – SA1-110 a new sorting algorithm for auto contact results has been introduced. Separate surfaces and contacts/ties were created if the contact surfaces between two components were not fully connected. Some of our users wanted the simplified sorting algorithm back. Thus, a new option is available to create only one contact pair between two components regardless of in how many places they are in proximity.

Midsurface Thickness Map Utility

Midsurface Thickness Map utility now can define thicknesses on nodes. When running this tool, the user automatically receives a *NODAL THICKNESS card with all nodes and thicknesses of the midsurface mesh listed under and the NODAL THICKNESS parameter on the property card.

---

*REBAR LAYER

The *REBAR LAYER card has been introduced to the Abaqus interface. The keyword is supported in the shell (Composite/Non-composite), membrane and surface section cards. A new checkbox ‘Rebar layer’ in the Composites > Material Orientation panel is available. If this box is checked a user can assign and review the material orientation of his rebar layer definition. In case of GEOMETRY=CONSTANT the rebar orientation angle can be reviewed for each individual layer (not implemented for ANGULAR or LIFT EQUATION geometry).

Resolved Issues

- For curve definitions (*AMPLITUDE) the reader can now distinguish between a definition with one or four pairs of x,y values per line.
- Beam masses are now calculated, even if no HyperBeam section is present.

- On import, for surface definitions on cohesive elements a warning message came up stating that some elements had invalid face identifiers. This has been resolved now.

Back to top

11.0.101 Release Notes - HyperView

Abaqus 6.10-EF and 6.11 Readers

The Abaqus 6.10-EF and 6.11 readers has been added for Windows 32/64bit and Linux 64bit. HyperWorks 11.0 supports also these versions: 6.8, 6.8-EF, 6.9, 6.9-EF, 6.10.

Sorting Algorithm for Auto Contact Results

The sorting algorithm for contact results has been improved and leads to better performance.

Back to top

Highlights of the ANSYS Solver Interface 12.0 Release

- Bar Panel Updates
- Supported Versions
- HyperView Interfacing New Features
- Resolved Issues

Contents

11.0.130 Release Notes

Ply Angles

Contact Manager and Auto Contact Enhancements

FLUID116 and LINK34 Elements

Resolved Issues

New Results Support (HyperView)

Resolved Issues (HyperView)

11.0.120 Release Notes

Support of ANSYS MPC Sections

New Elements Support

Resolved Issues

New Script for Calculating the Total Sum of Nodal Forces and Moments

11.0.101 Release Notes

New Version Support

New Elements

HyperLaminate Support of Composite Element Definition

New Gasket Element Support

New Version Pipe Elements

Coupling for Thermal Models

Resolved Issues

Support of ETABLE Outputs (HyperView)

Commonly Used Element Types Now Supported (HyperView)

Result Output Files (HyperView)

Vector Plots (HyperView)

New Elements Support (HyperView)

Known Limitations

12.0 Release Notes

Bar Panel Updates

Bar panel in 1-D page has been now customized for ANSYS profile. Options to create bar elements such as “component”, “vector” are not applicable for ANSYS solver and hence have been removed. Within ‘Plane’ and ‘Node’, only options relevant to ANSYS are retained. This helps user to pick the right option to create bar elements in HyperMesh.

Supported Versions

HyperMesh-Ansys interface is now updated to ANSYS14.0 release.

HyperView Interfacing New Features

- HV now supports ANSYS14.0 result files
- ETABLE results are now available for the following thermal elements:
  - SOLID70, SOLID87, SOLID90 and SURF152
- HV now supports ANSYS INFIN9 and INFIN110 element types

Resolved Issues

HyperMesh interface resolved issues:

- Issue with the export of load collectors by “displayed” option where all displayed collectors were not exported has been fixed

Back to top

11.0.130 Release Notes - HyperMesh

Ply Angles

Ply angles for each layer of composites defined by SHELL SECTION option can now be viewed through the Composite panel.

Contact Manager and Auto Contact Enhancements

- Options of selecting multiple rows of contact pairs to editing properties, materials and delete are now available in the Contact Manager. In earlier versions, you could select only one row to edit.
- Normals for contact surfaces are now auto-set by the Contact Wizard considering the positions of

master and slave. Previously, you needed to reset the normals after creating the pair. This feature is also in the Auto Contact tool.

FLUID116 and LINK34 Elements

FLUID116 and LINK34 elements have been supported under rods. Surface thermal loads can now be applied on these elements using the Pressure panel.

Resolved Issues

- The Auto Contact tool did not allow you to graphically select a component after the first time selection.
- Beam offsets are now considered in 3D beam visualization.

Back to top

11.0.130 Release Notes - HyperView

New Results Support

Results support for following elements have been added in ANSYS result reader:

- PLANE121, CIRCU124, CONTA177, PLANE223, SOLID226, SOLID227

Results support for following Gasket elements is now available:

- INTER192, INTER193, INTER194 and INTER19

Result type “Contact Forces” is now supported as Vector results.

Resolved Issues

- Issue with load case title for ANSYS 13 result files where HyperView displays the wrong title has been fixed.

Back to top

11.0.120 Release Notes - HyperMesh

Support of ANSYS MPC Sections

HyperMesh now supports ANSYS MPC sections - JOINTS. Subtypes such as Revolute, Universal, Slot, Point-to-plane, Revolute Translational, Cylindrical, Planar, Weld, Orient, Spherical, General and Screw are supported. Other joint section control cards such as SECJOINT, SECLOCK and SECSTOP are also now supported. All these joint section control cards are available under BEAMSECOL with type JOINT.

New Elements Support

New Elements Support: INFIN9 and INFIN110 are now supported in HyperMesh.

Resolved Issues

Issue with mass calculation for Mass21 element type is fixed.

Back to top

11.0.120 Release Notes - HyperView

New Script for Calculating the Total Sum of Nodal Forces and Moments

A new script has been added in HyperView to calculate the total sum of nodal forces and moments for selected nodes. This script can be accessed from the HyperView menu bar Tools > Extract summed node force/moments. This is a feature similar to "Print Summed elemental forces" or PRNLD command available in ANSYS post-processing.

Back to top

11.0.101 Release Notes - HyperMesh

New Version Support

HyperMesh now supports ANSYS 13.0

New Elements

The following new elements introduced in ANSYS 13.0 are now supported in HyperMesh:

- FLUID220
- FLUID221
- SOLID278
- SOLID279

HyperLaminate Support of Composite Element Definition

HyperLaminate now supports composite element definition for those elements where layer properties are defined using ANSYS section data (SECDATA card, TYPE-SHELL). Element types SHELL181, SOLID185, SOLID186, SOLSH190 and SHELL181 are the elements supported in this category.

New Gasket Element Support

Gasket elements now supported: INTER192, INTER193, INTER194, INTER195 and INTER205.

New Version Pipe Elements

New version pipe elements: PIPE288, PIPE289, ELBOW290, and SOLID285.

Coupling for Thermal Models

Coupling (CP) for thermal models has been added.

Resolved Issues

- Loads such as forces and moments applied on the same nodes but in different load collectors, if added in a load step, were exported without summing the load values. However, the ANSYS solver considers the last load applied at the node (data is appended with the latest value). This issue has been fixed so that the loads applied on the same nodes under a load step will be summed up.
- ANSYS arrays and tables are now supported as unsupported data. This ensures that these TABLES and arrays are included while exporting the model from HyperMesh.

Back to top

11.0.101 Release Notes - HyperView

Support of ETABLE Outputs

Support of ETABLE Outputs for commonly used ANSYS elements:

- Many result items use ETABLE, such as summable miscellaneous (SMICS), non-summable miscellaneous (NMISC), absolute value of variation of nodal stress component (SDSG), heat flow (HEAT), fluid flow (FLOW), element surface data value (SURF), or undeformed element centroidal location (CENT).
- These result items vary from element to element. You need to note the item name for each result output quantity and its sequence number from ANSYS documentation and then pick the same in HyperView under “Result Type” to post-process.

Commonly Used Element Types Now Supported

The following is a list of the most commonly used element types now supported: BEAM4, PLANE42, BEAM44, SOLID45, SHELL57, SHELL63, SHELL93, SOLID95, SHELL99, SHELL181, and BEAM188.

Result Output Files

HyperView now supports ANSYS’s latest version for result output files.

Vector Plots

Vector plots are now supported for element nodal forces and moments.

New Elements Support

Results support for the following elements have been added: FLUID220, FLUID221, SOLID278, SOLID279, PIPE288, PIPE289, ELBOW290, and SOLID285.

Known Limitations

Results for the following element types are not yet supported: Gasket elements – INTER192, INTER193, INTER194, INTER195 and INTER205.

Back to top

Highlights of the CAD Interfacing 12.0 Release

- General
- ACIS
- Catia
- DXF
- IGES
- JT
- Parasolid
- ProE
- SolidWorks
- STEP
- UG
- VDAFS

Contents

11.0.130 Release Notes

General

Catia

DXF

FiberSim

IGES

JT

Parasolid

ProE

SolidWorks

STEP

UG

VDAFS

11.0.120 Release Notes

General

ACIS

CATIA

IGES

Parasolid

ProE

SolidWorks

UG

11.0.101 Release Notes

CATIA

IGES

JT

STEP

UG

12.0 Release Notes

General

New Features/Enhancements

- Added @CreateMissingSurfaceContours option to generate contour of surface that failed importing due to cleanup issues
- CAD reader INI files are also looked for in HW_CONFIG_PATH defined directories
- Import of CAD axis systems as HyperMesh entities
- Improved error message when trying to import a CAD file with the wrong format

Resolved issues

- Removed incorrect error message about missing license file in license server log
- Enhanced import cleanup algorithm when very small surfaces are present in the model

---

ACIS

New Features/Enhancements

- ACIS R21 support

Resolved issues

- Fix missing surfaces at import for specific models

Catia

New Features/Enhancements

- CATIA V5-6R2012 support
- Added support for entity names in non-Latin alphabets

Resolved issues

- Improved import quality
- Fix surface normal orientation when importing CATIA files
- Fixed edge stitching issue when importing specific CATIA file

---

DXF

Resolved issues

- Fixed crash when importing specific files

IGES

New Features/Enhancements

- Surfaces get names at IGES export
- New @SkipConnectivityComputation allows quick IGEs import without connectivity information for review purposes

Resolved issues

- Fixed missing surfaces when importing specific files
- Fixed incorrect edges when importing specific IGES files
- Fixed HM hanging when importing specific files
- Fixed inconsistent surface normals at import time with specific models

---

JT

New features/Enhancements

- Added more details about JT assemblies import to jt_reader.msg file

Resolved issues

- Improved import quality and time
- Improved assembly reading time
- Fixed issue with importing spherical surfaces
- Fixed HyperMesh not responding when importing specific JT files
- Fixed bug preventing to import JT file after UG file has been imported
- Fixed issue with parts not being positioned correctly in specific assembly

---

Parasolid

New Features/Enhancements

- Added support for entity names in non-Latin alphabets

Resolved issues

- Fixed model specific import issues
- Fixed HyperMesh not responding when importing specific Parasolid files
- Fixed issues with component names on Linux
- Fixed issue with surface being split into many surfaces at export
- Fixed import issues with small scale models
- Fixed issues with surfaces projecting out at import time
- Fixed being unable to export STEP file to root directory on Windows platform

---

ProE

New Features/Enhancements

- Added support for entity names in non-Latin alphabets

Resolved Issues

- Fixed import issues for specific models
- Fixed import issues with assemblies located in root directories on Windows
- Fixed incorrect metadata at import

SolidWorks

New Features/Enhancements

- SolidWorks 2012 support
- Added support for entity names in non-Latin alphabets

Resolved issues

- Fixed issue with incorrect component names on Linux
- Fixed issues with importing assemblies on Linux

STEP

New Features/Enhancements

- Added support for entity names in non-Latin alphabets

Resolved issues

- Fixed issue with solids lost at import
- Fixed issue with metadata lost at import
- Fixed reading of composite curves
- Fixed being unable to export STEP file to root directory on Windows platform

UG

New Features/Enhancements

- UG NX 8.0 support is now available on all supported platforms
- Import of UG/NX files without requiring NX being installed

Resolved Issues

- Fixed issue with UG weld data
- Fixed HM hanging when importing specific model
- Fixed issues with missing surfaces at import

VDAFS

Resolved Issues

- Fixed HM hanging when importing specific model

Back to top

11.0.130 Release Notes

General

New Features/Enhancements

- Option to split imported CAD files into component by bodies is now available
- Option to import color metadata as real values has been added for ProE and STEP formats.
- Option to keep original CAD files lines breakup by not merging them is now available
- Option to optimize exported files for CAD now available directly from the Export panel.

Resolved Issues

- Removed incorrect error message about missing license file in license server log.

Catia

New Features/Enhancements

- List missing files if any occur when importing a CATIA CATProduct file

Resolved Issues

- Fixed wrong surfaces with some models during CATIA import
- Fixed FULL_IDENTIFIER metadata being incomplete
- Correctly take hidden parts information in HM ExtAPI during import

DXF

Resolved Issues

- Resolved issues with DXF import

FiberSim

FiberSim HDF5 reader has been added to directly read Composites Design data from FiberSim. This reader can be accessed in the Cad readers import option. The following information can be read from FiberSim HDF5 files with composite data:

- Plies- Name, thickness, and fiber orientation information is directly read and mapped as Ply entity in HyperMesh. Plies that point to 'woven' and 'stack' materials are split/separated into multiple plies with half the thickness and correct orientation angles. These split ply names always have a '_1' and '_2' suffix for easy identification.
- Data Map/Table – Data map with element set (ply shapes), material orientation angles (orient1, orient2, draping corrections) thickness corrections, reference direction and normal information for every FiberSim element of each ply is preserved/mapped in the Table entity in HyperMesh. Hence, each ply has a table associated with it. HyperMesh does not create nodes and elements in database from FiberSim triangulation data to define ply shapes. Instead, it preserves this information in a table so that when these plies are mapped (realized) on actual good mesh, HyperMesh uses this triangular information to define the ply boundary and extract the actual elements.
- Laminates – One laminate per HDF5 component with all the ply sequence preserved as per 'layer_id' value.
- Materials – Material names and its mechanical properties are read and mapped to solver cards automatically, depending on the user profile loaded while importing the model. Currently mechanical properties such as, E1, E2, E3 , G12,G13,G23, Alpha1, Alpha2 and Alpha_ref temperature are mapped to solver material attributes.
- Rosette/Systems – All the system definitions available in HDF5 file will be imported into one system collector in HyperMesh. Currently HyperMesh does not preserve the ply and system relation.

Once imported, the data map (FiberSim triangulation element information) can be realized into a finite element representation. In this process elements, are associated with ply shapes previously defined through FiberSim triangulation. Along with this, drape data is also mapped to the new ply elements.

IGES

New Features/Enhancements

- Solid import for IGES format is now available

Resolved Issues

- Improved support for IGES types 192 and 196
- Improved reading of anisotropic surfaces

JT

Resolved Issues

- Fixed wrong surfaces with some models during JT import

Parasolid

Resolved Issues

- Fixed export of specific models not exporting as Parasolid.
- Fixed elliptical surfaces being exported as flat surfaces.
- Display correct message when trying to export an empty file as Parasolid.
- Improved geometry quality with Parasolid import.
- Improved performances for Parasolid import

ProE

Resolved Issues

- Weld information reading improvement.
- Fixed missing surfaces when importing some models

SolidWorks

Resolved Issues

- Improved geometry quality with SolidWorks import

STEP

Resolved Issues

- Import of names of edges, faces, points, and free-line as metadata is now available

---

UG

New Features/Enhancements

- UG NX 8.0 support is now available.

Resolved Issues

- Resolved issue with weld data import.
- Fixed import for some specific models that did not import

VDAFS

Resolved Issues

- Fixed crash with specific models during VDAFS import

Back to top

11.0.120 Release Notes

General

Resolved Issues

- Proper import of solids with inner cavities

ACIS

Resolved Issues

- Resolved several issues with missing or corrupt surfaces after import
- Resolved several issues with files not importing

CATIA

New Features/Enhancements

- CATIA reader has been enhanced to read directly Composites Parts Design (CPD) data. If the new option 'Import composites data' is enabled in the Import geometry tab, the following information can be read from Catia files with composite data:
  - Ply shape: Lines representing the geometric ply shape.
  - Ply data, such as name, thickness, material, relative angles
  - System definitions
  - Stacking information on the ply order is imported into the HM laminate entity
  - Materials
- Once imported the geometric information can be realized into a finite element representation. In this process elements of a shell or continuum solid mesh are associated with ply shapes previously defined through lines.
- Please find further information on this new process in the plies and laminate documentation.
- Added support for CATIA V5R21 files at import.
- Resolved several issues with missing or corrupt surfaces after import.

IGES

Resolved Issues

- Resolved several issues with missing or corrupt surfaces after import

---

Parasolid

New Features/Enhancements

- Added support for importing face and solid names as metadata

Resolved Issues

- Resolved issue with solids not created properly upon import.
- Resolved several issues with missing or corrupt surfaces after import

ProE

New Features/Enhancements

- Added assembly level features support

Resolved Issues

- Resolved several issues with missing or corrupt surfaces after import
- Resolved several issues with files not importing

SolidWorks

New Features/Enhancements

- Added SolidWorks 2011 support

UG

Resolved Issues

- Resolved issue with solids not created properly upon import

Back to top

11.0.101 Release Notes

CATIA

Resolved Issues

- Now able to properly read CATIA V5R20 files from all service packs and hotfixes
- Resolved several issues with missing or corrupt surfaces after import

---

IGES

Resolved Issues

- Resolved several issues with missing or corrupt surfaces after import

---

JT

Resolved Issues

- Resolved several issues with missing or corrupt surfaces after import

---

STEP

New Features/Enhancements

- Ability to export geometry to STEP AP214.

Resolved Issues

- Resolved several issues with color metadata not being read correctly

UG

New Features/Enhancements

- Added support for UG NX7.5 on Win32, Win64 and Linux64.
- Note: The UG NX7.5 UGOpen libraries require the Microsoft Visual C++ 2008 Redistributable Package to be installed on each Windows client machine. This is not provided with HW11.0.101-HWDesktop, but should be provided with a valid UG installation. This is also available from the Microsoft website.

Resolved Issues

- Edge stitching is improved, resulting in fewer free edges after import for some models
- Resolved issue with some models containing mid-surfacing information that did not import correctly

Back to top

Highlights of the LS-DYNA Solver Interface 12.0 Release

- Support of *PARAMETER and *PARAMETER_EXPRESSION
- Performance and Speed Improvements
- Solver Browser
- Maximum Number of Characters
- Keyword Updates
- New HyperView Interfacing Features
- Resolved Issues

Contents

11.0.130 Release Notes

Box Trim

Solver Browser

Updated Keywords

HM Assembly Browser Creation Options

ID Pools Added

I/O Support for PGP Encryption

Resolved Issues

New Data TypE (HyperView)

Combination of Results (HyperView)

CPM Results (HyperView)

New Reader Options (HyperView)

11.0.120 Release Notes

*INCLUDE_PATH

Elements and Properties Conversion

Resolved Issues

D3PLOT Reader (HyperView)

FEMZIP LS-DYNA Reader (HyperView)

LS-DYNA HyperGraph Readers (HyperGraph)

11.0.101 Release Notes

Transformations in the Model

*INCLUDE STAMPED_PART and *INCLUDE_STAMPED_PART_SET

Import Browser

Export Enhancements
Keyword Support
New Cards
*INCLUDE_TRANSFORM_PATH
*MASS_PART_SET
*DEFINE_COORDINATE_NODE
*CONSTRAINED_LAGRANGE_IN_SOLID
*ALE_REFERENCE_SYSTEM_GROUP
*INITIAL_VOLUME_FRACTION_GEOMETRY
*CONSTRAINED_NODAL_RIGID_BODY
ID Pools
Resolved Issues
*INCLUDE_TRANSFORM
Maximum Limit on Number of D3PLOT Files
2nd Order Tetra Elements
FEMZIP V 5.49
*AIRBAG_PARTICLES

12.0 Release Notes

Support of *PARAMETER and *PARAMETER_EXPRESSION

- *PARAMETER and *PARAMETER_EXPRESSION support is limited to Input/output only for the keyword mapped to the following HM entities: components, materials, properties, curves, loads, load collectors, system collectors, and control volumes.
- Parameterization of entity IDs not supported
- Unsupported parametrized entities are replaced with corresponding values during import to avoid loss of data.
- The variables parametrized will be grayed out in the card editor so that it can’t be changed.
- In macros and browsers, if any parameterized variable is changed a warning message appears in the message bar and the value will not be changed.

Performance and Speed Improvements

- Performance/speed improvement in Nastran to LS-DYNA conversion
- Performance/speed improvement in import/export of large models

---

Solver Browser

Show/Hide/Isolate functionalities for Include files in the Solver Browser are now available.

Maximum Number of Characters

Maximum number characters for names of components, materials, and properties extended to 80 in import and export.

Keyword Updates

Updates to the following keywords:

- *MAT_ENCHANCED_COMPOSITE_DAMAGE_TITLE
- *CONTROL_SPOTWELD_BEAM
- *CONSTRAINED_EXTRA_NODES_NODE/SET
- *CONTROL_IMPLICT_FORMING
- *INITIAL_VELOCITY_RIGID_BODY
- *MAT_FU_CHANG
- *MAT_LOW_DENSITY_SYNTHETIC_FOAM
- *CONTROL_CONTACT

New HyperView Interfacing Features

- Support of d3int files
- FEMZIP reader updated to FEMZIP libraries v 6.0
- New FEMZIP reader options added
  - Effective strain with prestrain removed – Removes the effective strain value at time 0 from all the time steps.
  - Strain Tensor with prestrain removed – Removes the strain tensor value for all its components at time 0 from all the time steps.
- Effective plastic strain at mid layer added for shell elements if it is missing in the file.

Resolved Issues

The following HyperMesh interfacing issues have been resolved:

- Component Table and Model Browser synced to reflect the changes in thickness and mass
- Issue with reading *BOUNDARY_SPC_NODE_ID is fixed.
- Export of *INCLUDE_STAMPED_PART and *INCLUDE_STAMPED_PART_SET at the end of the deck.
- Issues with reading *DEFINE_TABLE is fixed.
- Issue with reading Nastran file into LS-DYNA user profile fixed
- Reading issue with *CONTACT when either slave or master not defined is fixed.
- Reader improved to handle decks with multiple *INCLUDE_STAMPED_PART and *INCLUDE_STAMPED_PART_SET
- Improved reading of *MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC_ECHANGE
- Issue with export of *CONTACT_RIGID_SURFACE addressed.

The following HyperView interfacing issues have been resolved:

- Model reader fixed to handle input decks with *INCLUDE_STAMPED_PART, *INCLUDE_ STAMPED_PART_SET, and *INCLUDE_TRANSFORM
- Crash of HyperView when animating with contour on result files with airbag particles fixed

Back to top

11.0.130 Release Notes - HyperMesh

Box Trim

Box trim feature was updated to handle the cleanup of FE entities in the model after the trimming process to result in error free solver deck.

Solver Browser

The Solver Browser Show/Hide/Isolate/Review functions were updated for better performance. Review will display them in its color, if a color has been defined. If not, the display is red.

Updated Keywords

Updates have been made to following keywords:

- *MAT_GURSON_JC, *MAT_033, *MAT_103
- *DATABASE_BINARY_RUNRSF
- Optional card D for *Contact
- *BOUNDARY_SPC_BIRTH_DEATH

HM Assembly Browser Creation Options

The option to create an HM Assembly based on includes structure during import is now available.

ID Pools Added

ID pools added to the load collector entity now allows duplicate IDs among its mapped keywords in the LS-DYNA user profile. A new load collector with no card image will be created automatically if the current load collector has a card image and you create any loads.

I/O Support for PGP Encryption

I/O support for PGP Encryption of selected materials:

- *MAT_ELASTIC, *MAT_RIGID, *MAT_SPOTWELD, and *MAT_NULL
- *MAT_SPRING_ELASTIC, and *MAT_FU_CHANG_FOAM
- *MAT_PIECEWISE_LINEAR_PLASTICITY

Resolved Issues

- Unresolved IDs in `.hm` files takes a long time to read. This is now quicker.
- Support of advanced sets for *BOUNDARY_SPC_SET is now available.

- *MAT_BARLAT_YLD2000 was missing a line – this has been added.
- The error in export of IDs of FAMMG and BAMMG referred in *INITIAL_VOLUME_FRACTION GEOMETRY has been repaired.
- Reader crash during unsupported control card import has been fixed.
- Reader crash with duplicate control cards no longer occurs.
- Error in reading *CONTROL_SPOTWELD_BEAM due to lack of support has been fixed.

Back to top

11.0.130 Release Notes - HyperView

New Data Type

Support of new data type Dtemperature/DTime has been introduced.

Combination of Results

The combination of Beam results, Mass scaling results and Thermal results is now supported.

CPM Results

Support all formats of Corpuscular Particle Method (CPM) results is available.

New Reader Options

Introduced new reader options to improve the performance of commonly performed math operations:

- Strain Tensor with Prestrain removed – Subtracts the strain Tensor of first step from all time steps for all elements.
- Effective plastic strain with Prestrain removed – Subtracts the effective plastic strain of first step from all time steps for all elements.
- Maximum effective plastic strain over the layers – Gets the maximum effective plastic strain over the layer/thickness of shell elements.
- Maximum effective plastic strain over the layers with Prestrain removed – Subtracts the effective plastic strain of first step from all time steps then gets the maximum of over the layer/thickness of shell elements.

Back to top

11.0.120 Release Notes - HyperMesh

*INCLUDE_PATH

*INCLUDE_PATH support updated to handle nested include files.

Elements and Properties Conversion

1D elements (CBEAM, CBAR, CROD, CBUSH, CELAS, and RBE3) and their properties (PBEAM, PBAR, PROD, and PBUSH) are converted to equivalent element, property and material in LS-DYNA.

Mass element COMN2 is converted to *ELEMENT_MASS or *ELEMENT_INERTIA in LS-DYNA.

Resolved Issues

- Reader to updated to handle unsupported keyword *AIRBAG_SHELL_REFERENCE_GEOMETRY, *CONTROL_MPP_DECOMPOSTION_DISTRIBUTE_ALE_ELEMENTS and loss of shell elements with thickness values defined.
- Transformation Manager updated to handle includes with long path.
- Renumbering of set ID's upon reading legacy HM files is fixed.

Back to top

11.0.120 Release Notes - HyperView/HyperGraph

D3PLOT Reader

D3PLOT reader updated to handle names for components introduced in LS-DYNA V971 R 5.1.1.

FEMZIP LS-DYNA Reader

FEMZIP LS-DYNA reader is reverted back to HW 11.0 status FEMZIP version 5.43, due to issues with FEMZIP libraries 5.49 from Fraunhofer.

LS-DYNA HyperGraph Readers

LS-DYNA HyperGraph readers are updated to handle session files with different settings for the environment variable DYNA_REQUEST_TYPE.

Back to top

11.0.101 Release Notes - HyperMesh

Transformations in the Model

Transformations defined in the model by default will be turned on during import.

*INCLUDE STAMPED_PART and *INCLUDE_STAMPED_PART_SET

Files referred using keywords *INCLUDE_STAMPED_PART and *INCLUDE_STAMPED_PART_SET are by default imported as read only into HyperMesh to preserve the associativity.

Import Browser

In the solver options of the Import Browser, an option to choose the type of the Include file is now available at the time of importing a file as include to the file in session. This allows the user to set the include file type as one of the following: INCLUDE, INCLUDE_STAMPED_PART, INCLUDE_STAMPED_PART_SET, and INCLUDE_COMPENSATION_options.

During import of .k files from 11.0 or older versions, the user can choose between HM name or name in Keyword using the check box option "Map LS-DYNA TITLE/HEADING to HM name" in the solver options of Import Browser.

Export Enhancements

New check box introduced in the Export tab to avoid exporting any HM comments into the deck.

During export, comment line HM name will only appear if the keyword does not support name.

Keyword Support

Support of the following keywords is reorganized for better user interaction.

- DMPING_PART_MASS moved from HM entity component to Property
- DAMPING_PART_STIFFENESS moved from HM entity component to Property.
- INCLUDE_STAMPED_PART moved from Component card image to Include file.

LS-DYNA keywords mapped to the named HM entities such as assembly, component, properties, materials, groups, sensors, control volume, load collectors, system collectors, vectors collectors, curves, plots, etc. will have the name given at the time of creation mapped to the name available in its card image if the ID or TITLE option for the keywords is turned on. This ensures maintaining one name per keyword/entity and facilitates renaming from browser as well as panels.

New Cards

The following new cards have been added new for LS-DYNA 971 user profile:

- *DAMPING_PART_MASS_SET – Supported as Property
- *DAMPING_PART_STIFFNESS_SET – Supported as Property
- *INCLUDE_STAMPED_PART_SET - Supported as Include file
- *PART_COMPOSITE_TSHELL – Supported a component
- *ELEMENT_SHELL_COMPOSITE – Supported as QUAD and TRIA element
- *CONSTRAINED_JOINT_RACK_AND_PINION_ID – Supported as FEJOINTS
- *CONSTRAINED_JOINT_GEARS – Supported as FEJOINTS
- *CONSTRAINED_JOINT_TRANSLATIONAL_MOTOR – Supported as FEJOINTS
- *CONSTRAINED_JOINT_ROTATIONAL_MOTOR – Supported as FEJOINTS
- *CONSTRAINED_JOINT_CONSTANT_VELOCITY – Supported as FEJOINTS
- *CONSTRAINED_JOINT_PULLY – Supported as FEJOINTS
- *MAT_MOMENT_CURVATURE_BEAM – Supported as FEJOINTS
- *MAT_COHESIVE_MIXED_MODE – Supported as FEJOINTS.

***INCLUDE_TRANSFORM_PATH**

In the past the keyword *INCLUDE_TRANSFORM_PATH was supported for I/O only. Now user can associated include path to any include file using the Include file options in the Include Browser.

***MASS_PART_SET**

*MASS_PART_SET supports its definition on *SET_NODE_GENERATE.

***DEFINE_COORDINATE_NODE**

*DEFINE_COORDINATE_NODE was added support for variable DIR.

***CONSTRAINED_LAGRANGE_IN_SOLID**

*CONSTRAINED_LAGRANGE_IN_SOLID was updated to LS-DYNA 971 R 5.1.

***ALE_REFERENCE_SYSTEM_GROUP**

*ALE_REFERENCE_SYSTEM_GROUP was updated to LS-DYNA 971 R 5.1.

***INITIAL_VOLUME_FRACTION_GEOMETRY**

*INITIAL_VOLUME_FRACTION_GEOMETRY was updated to LS-DYNA 971 R 5.1.

***CONSTRAINED_NODAL_RIGID_BODY**

Support of the keyword *CONSTRAINED_NODAL_RIGID_BODY is updated to allow its definition on complex sets such as *SET_NODE_ADD, *SET_NODE_ADD_ADVANCED, and *SET_NODE_GENERAL.

ID Pools

ID pools added to set entity to have duplicate ID's among its mapped keywords in LS-DYNA UP namely *SET_NODE, *SET_SHELL,*SET_SOLID,*SET_BEAM, *SET_DISCRETE, *SET_PART, *SET_SEGMENT_GENERAL.

ID pools added to sensor entity to have duplicate ID's among its mapped keywords in LS-DYNA UP namely *SENSOR_CONTROL, *SENSOR_DEFINE_CALC-MATH, *SENSOR_DEFINE_ELEMENT, *SENSOR_DEFINE_FORCE, *SENSOR_DEFINE_NODE, *SENSOR_SWITCH, *SENSOR_SWITCH_CALC-LOGIC.

ID pools added to control volume entity to have duplicate ID's among its mapped keywords in LS-DYNA UP namely *AIRBAG_options, *AIBAG_ADVANCED_ALE, *AIRBAG_ALE, *AIRBAG_INTERACTION, *AIRBAG_REFERENCE_GEOMETRY, *DEFINE_ALEBAG_BAG, *DEFINE_ALEBAG_HOLE, *DEFINE_ALEBAG_INFLATOR.

Resolved Issues

- Error in reading include files defined using INCLUDE_PATH fixed.
- Error in reading *CONSTRAINED_SHEEL_TO_SOLID fixed.
- Missing node sets during export from .hm files fixed.
- Error in mass calculation involving *ELEMEN_SEATBELT_ACCELEROMETER fixed.

Back to top

11.0.101 Release Notes - HyperView

***INCLUDE_TRANSFORM**

Supports reading model file with *INCLUDE_TRANSFORM.

Maximum Limit on Number of D3PLOT Files

Maximum limit on number of D3PLOT files read in HV session extended to 99999.

2nd Order Tetra Elements

Support of 2nd order Tetra elements has been added.

FEMZIP V 5.49

Updated to FEMZIP V 5.49 and performance improved.

*AIRBAG_PARTICLES

Support of *AIRBAG_PARTICLE results.

Back to top

Highlights of the Marc Interfacing 12.0 Release

- HyperView Interfacing Updates

Contents

11.0.130 Release Notes

Resolved Issues

Supported Elements (HyperView)

11.0.101 Release Notes

TRANSFORMATION Card

New Elements

FIXED TEMPERATURE

Enhancements to the Marc Template

Resolved Issues

12.0 Release Notes

HyperView Interfacing Updates

HV now supports following result types for Marc t16 files – Tying forces, Failure indices, Interlaminar shear & normal stresses, Stresses in preferred direction and Elastic strain energy density.

Back to top

11.0.130 Release Notes - HyperMesh

Resolved Issues

- Issue with element and node input listing where HyperMesh exports element/node lists in ranges using “TO” “AND” has been fixed.

Back to top

11.0.130 Release Notes - HyperView

Supported Elements

HyperView now supports the following Marc element types:

- Element5, Element26 and Element28.

Back to top

11.0.101 Release Notes - HyperMesh

TRANSFORMATION Card

You can click on Systems > Assign > Nodes and use the Set Displacement option in HyperMesh to assign a system to the nodes and export with the TRANSFORMATION card. Importing a model with the TRANSFORMATION card leads to "set displacement" for the nodes mentioned in the TRANSFORMATION card.

New Elements

ELEMENT28, ELEMENT149 has been added under the Marc profile.

FIXED TEMPERATURE

The temperature load FIXED TEMPERATURE card is now supported in the Temperature panel.

Enhancements to the Marc Template

The Marc template now exports element or node lists in ranges instead of exporting individual IDs. This will lead to the compact deck and reduce the file size. This has been implemented for nodes and element lists referred to sets, properties, loads and materials.

Resolved Issues

- Marc template for RBE2 defined with slave nodes more than 10 numbers was having issues with export formats. This has been corrected.
- Export issues with material type FOAM where data defined with multiple curves was not being exported are now fixed.

Back to top

Highlights of the Nastran Solver Interface 12.0 Release

- New Export Solver Option: Auto-adjust RBE3 0.0 weight factor
- New OP2 Reader Option: Deformed Shape: Allow all nodal vector result types
- OP2 Format Results File
- Resolved Issues

Contents

11.0.130 Release Notes

SOL 400 Support

New Property Type

Added TABLEM* to TABLE Create Macro

Added Support for Acoustic Conjugate Infinite Elements and Property

Support for 2D Element Forces Tensor Plot (HyperView/ HyperGraph)

11.0.120 Release Notes

Non-structural Mass added to Mass Calc

Auto NSMADD Added to Export Solver Options

Importing Grids with a Reference System in the Database

NVH Enhancements

Resolved Issues

Added Rotor Dynamics Animation Mode

11.0.101 Release Notes

SOL159 (Nonlinear Transient coupled with Heat Transfer) Support

New NX Keywords

STATSUB Additions

Filter by Card Image

Resolved Issues

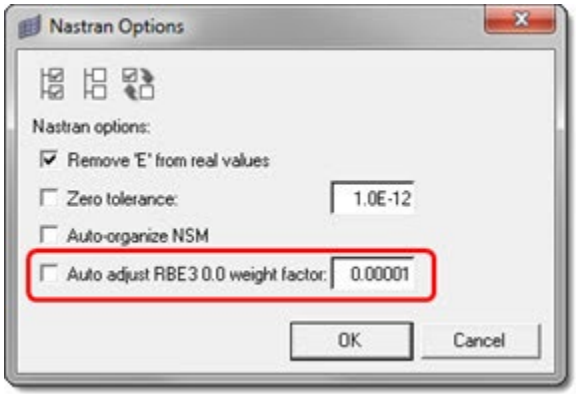
OP2 Reader Options (HyperView)

XDB Reader Options

12.0 Release Notes

New Export Solver Option: Auto-adjust RBE3 0.0 weight factor

When turned ON, HyperMesh will change any 0.0 weight factors on RBE3 elements to the specified small value. This adjustment is done in the exported solver deck only, and not in the HyperMesh model. This option is OFF by default.

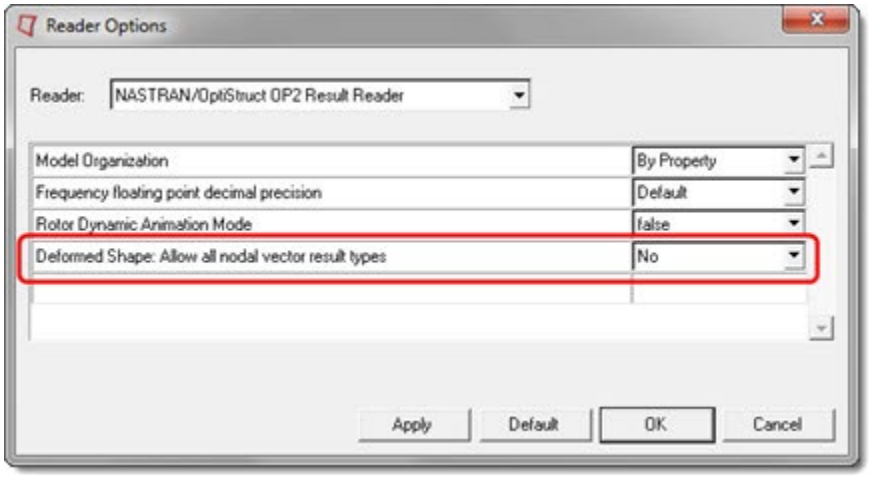


Nastran Model Checker

Additional checks have been added to the Nastran Model Checker.

New OP2 Reader Option: Deformed Shape: Allow all nodal vector result types

When switched to YES, this option allows all available nodal vector result types in the OP2 file to be used in the Deformed Shape panel. When switched to NO (default), the Deformed Shape panel will automatically filter result type selection according to analysis type.



OP2 Format Results File

HyperView now supports importing of CONM2 elements from OP2 format results file

---

Resolved Issues

HyperMesh interfacing resolved issues:

- NSML1 mass incorrectly distributed amongst 2D elements and properties issue has been corrected. This fix includes corrections to mass calc panel and summary templates.

- Pressure direction miss-assignment when importing PLOAD cards from separate files into a current model has been corrected.
- DRESP2 cards containing a large number of DTABLE entries missing after export issue has been corrected.
- TABLEx export in Long Format missing second line (empty) issue has been fixed.
- Import error due to keyword alignment of INIPENE option inside BCTPARM has been fixed.
- TABLED missing when using Export > Displayed option issue has been fixed.
- A memory issues causing BSURFs to be omitted on import has been corrected.
- DAREA/DELAY/DPHASE on SPOINTS not importing properly issue has been fixed.

HyperView interfacing resolved issues:

- An issue when plotting SDISP in result files containing both DISP and SDISP results has been resolved.

Back to top

11.0.130 Release Notes - HyperMesh

SOL 400 Support

SOL 400 support is now available. The 1st release of SOL400 support in HyperMesh introducing 50+ new cards and options, including:

- ANALYSIS = HSTAT / HTRAN for steady state and transient heat transfer.
- New Loadstep options for IC and NLSTEP.
- PARAM,TABS.
- PARAM,SIGMA.
- NLMOPTS: new control card.
- NLSTEP: new load collector.
- MATT4 as an extension to MAT4.
- PSHLN1 as an extension to PSHELL.
- PSLDN1 as an extension to PSOLID.
- RADM: new group type RADIATION.
- VIEW: new load collector.
- RADCAV: new load collector.
- VIEW3D as an extension to RADCAV.
- HEAT option inside BCBODY
- 15 new options inside BCTABLE.
- Automatic front/back SIDE assignment for CHBDYE on shells based on element normal for CONVECTION and RADIATION.
- Partial support of CHBDYG elements (import, edit, export only).

New Property Type

Added property type to NSM mass connectors. It is available for creation and FE absorption.

Added TABLEM* to TABLE Create Macro

- TABLEM1, TABLEM2, TABLEM3, TABLEM4

Added Support for Acoustic Conjugate Infinite Elements and Property

- CACINF3: New tria3 element
- CACINF4: New quad4 element
- PACINF: New 2D property with ACOUSTIC ID pool

Back to top

11.0.130 Release Notes - HyperView/HyperGraph

Support for 2D Element Forces Tensor Plot

- Plot and transform forces using the normal drawing mode [Fx, Fy, Fxy, Qy and Qz].
- Moment drawing mode available to visualize moments [Mx, My and Mxy].
- `.op2` file only.

Back to top

11.0.120 Release Notes - HyperMesh

Non-structural Mass added to Mass Calc

NSM1 and NSML1.

Auto NSMADD Added to Export Solver Options

Controls the grouping of all NSM* entities in the model and references them to case control on export.

Importing Grids with a Reference System in the Database

If a GRID card has a CP defined but no matching coordinate system defined in the input file, HyperMesh will now search the database for that existing system.

NVH Enhancements

- Added EIGRA as a loadcollector. Referenced by the METHOD case control selector.
- Added H3DCDS and EXCINP options to ASSIGN.

Resolved Issues

- RADIOSS (Bulk Format) PBUSH real number fields missing “E” reader bug is fixed.

- `cmd.exe` running in the background after launching the solver from HyperMesh issue is fixed.

Back to top

11.0.120 Release Notes - HyperView/HyperGraph

Added Rotor Dynamics Animation Mode

- Applies animation interpolation of mag*cos(wt+phi)
- Set in Nastran/OptiStruct OP2 Result Reader Options

Back to top

11.0.101 Release Notes - HyperMesh

SOL159 (Nonlinear Transient coupled with Heat Transfer) Support

SOL 159: Supported as a control card.

- ANALYSIS=HEAT: Supported in Global Case Control as a control card and within the Loadstep Editor.
- TSTEPNL(CC): Supported in Global Case Control as a control card and within the Loadstep Editor.
- FLUX: Supported as a Global Output Request control card and within the Loadstep Editor.
- TSTEPNL(BD): Supported as a load collector, and is referenced by Case Control command TSTEPNL.
- TEMPBC: Supported as a temperature load, and is referenced by the Case Control command SPC.
- TEMP: Supported as a temperature load, and is referenced by the Case Control command TEMP for static loadsteps, and is referenced in the LSEQ entry for dynamic loadsteps

---

New NX Keywords

- BCRPARA: Added keyword BCRPARA in loadcollectors.
- BCTSET: Added keyword BCTSET in Interfaces.
- BCTPARM: Added keyword BCTPARM as groups under BCTSET.
- BOLT: Supported as an entity set.
- BOLTFOR: Supported as a flux load.

STATSUB Additions

Added buckling and preload options in the loadsteps panel

- STATSUB(PRELOAD) and STATSUB(BUCKLING)

Filter by Card Image

Available in advanced entity selection panels for properties, materials and load collectors.

Resolved Issues

Load collectors without SID: Instead of placing cards without an SID into “misc*” load collectors on import, HyperMesh will give a hardcoded load collector name to each of these. For example, all ASET cards with DOF 123 will be collected into a load collector named “ASET_123.”

Back to top

11.0.101 Release Notes - HyperView/HyperGraph

OP2 Reader Options

These new reader options will allow users to control the way HyperView creates components on import.

- By Property: Creates components based on property IDs within the op2 file
- By Element Dimensions: Grouping based on 0D, 1D, 2D and 3D
- By 1 Component: Creates only 1 component

XDB Reader Options

These new reader options will allow users to control the way HyperView creates components on import.

- By Property: Creates components based on property IDs within the xdb file.
- By SuperElement / 1 Component (No SuperElement): Creates and organizes components by super element. If no super elements exist, all elements are grouped into 1 component.

Back to top

Highlights of the PAM-CRASH Solver Interface 12.0 Release

- Supported Versions
- Supported Keywords Update - PAM 2009 User Profile
- Supported Keywords Update - PAM 2010 User Profile
- Newly Supported Keywords - PAM 2010 User Profile
- ID Pools
- New HyperView Interfacing Features
- Resolved Issues

Contents

11.0.130 Release Notes

PAM 2010

Updated Keywords for PAM 2009

Resolved Issues

11.0.120 Release Notes

Added Rotor Dynamics Animation Mode (HyperView)

DSY FEMZIP Reader (HyperView)

PAM 2011 Support (HyperView)

11.0.101 Release Notes

PAM 2009

PAM 2008 Material Keywords Updated

PAM 2008 Boundary Conditions Keywords Updated

PLINK, LLINK Connector Keywords

Resolved Issues

Support of PAM2010

FEMZIP Reader

12.0 Release Notes

Supported Versions

Supported versions PAM 2G 2005 – PAM2G 2010

Supported Keywords Update - PAM 2009 User Profile

- CHAMBER
- CONLO
- ECTRL
- FRAME
- FUNCTION_SWITCH
- GROUP
- OCTRL
- MAT16, MAT214
- DELEM

Supported Keywords Update - PAM 2010 User Profile

- MAT105, MAT 130/131/132
- UNIDIRECTIONAL_COMPOSITE
- ERFOUTPUT
- CHAMBER
- MASS
- SHELL TIME STEP

Newly Supported Keywords - PAM 2010 User Profile

- FABRIC COMPOSITE B-PHASE PLY MODEL
- TETRA10
- BDFORC
- FPMCONTROL
- OTMCO

ID Pools

ID pools added for sensor entity. Keywords DELEM, FUNCTION_SWITCH, and SENSOR are placed in 3 different ID pools to allow duplicate ID’s as per solver.

ID pools added for property entity in PAM user profile. Each keyword is now mapped to a separate ID pool.

New HyperView Interfacing Features

- Supported versions PAM 2G 2005 – PAM2G 2011
- Support of PAM 2011 results in ERHF5 files

Resolved Issues

- Issue with export of MAT 143 fixed
- Issue with Rigid element creation and connectivity update when selecting sets with more than 10000 nodes from rigids panel fixed

Back to top

11.0.130 Release Notes - HyperMesh

PAM 2010

PAM 2010 user profile has been introduced to support I/O. New keywords specific to PAM 2010 have been added.

Updated Keywords for PAM 2009

The following keywords are updated in PAM 2009 user profile:

- Contacts: TYPE 37, TYPE 44, and TYPE 46
- Materials: MAT 37, MAT 213, MAT 223, and MAT 224
- Others: CDATA, RUPTURE, FRICTION, SENSOR, PART_BEAM, and HEAT

Resolved Issues

Error in slave node count in RBODY manager is now fixed.

Back to top

11.0.120 Release Notes - HyperView/HyperGraph

Added Rotor Dynamics Animation Mode

- Applies animation interpolation of mag*cos(wt+phi).
- Set in Nastran/OptiStruct OP2 Result Reader Options

DSY FEMZIP Reader

DSY FEMZIP reader updated to handle files with eroded elements.

PAM 2011 Support

Support of PAM 2011 results in the file formats .DSY, and .THP.

Back to top

11.0.101 Release Notes - HyperMesh

PAM 2009

Introduction of PAM2009 User Profile

PAM 2008 Material Keywords Updated

- MATL_12
- MATL_14
- MATL_15
- MATL_19
- MATL_24
- MATL_28
- MATL_35
- MATL_38
- MATL_51
- MATL_152
- MATL_214
- MATL_225
- PLY

PAM 2008 Boundary Conditions Keywords Updated

PAM 2008 Boundary Conditions keywords updated include TEMBC, INTEM and CONLO.

PLINK, LLINK Connector Keywords

PLINK, LLINK connector keywords have been updated to handle a maximum of 15 components in its card image. GES will be created if it has more than 15 components.

Resolved Issues

- Error in reading global variables has been fixed
- Error in I/O of CNTACT 36 is fixed

Back to top

11.0.101 Release Notes - HyperView

Support of PAM2010

Support of PAM2010 results in HDF5 file format (.ERHF).

FEMZIP Reader

FEMZIP Reader supports all the results from PAM 200 and is updated to FEMZIO V5.49. Performance is improved.

Back to top

Highlights of the RADIOSS Block Interface 12.0 Release

- Supported Versions
- New Keyword Support
- Resolved Issues

Contents

11.0.130 Release Notes

Solver Browser

INISHE/EPSP and INISHE/EPSP_F

New Keywords

Keywords Updated

Resolved Issues

New Reader Options (HyperGraph)

Resolved Issues (HyperView)

Resolved Issues (HyperGraph)

11.0.120 Release Notes

Keyword Support

Resolved Issues

HVTrans

11.0.101 Release Notes

Transformations

Export Tab Enhancements

ACCEL

Newly Supported Keywords

Updated Keywords

ID Pools

Resolved Issues

Creation of Section Cuts

Resolved Issues

12.0 Release Notes

Supported Versions

Supported versions RADIOSS 44, 51,90, 100 and 110

New Keyword Support

- INISHE/INISH3 OTHRO
- INISHE/INISH3 OTHRO_LOC
- INISHE/INISH3 EPSP
- INISHE/INISH3 EPSP_F
- INISHE/INISH3 STRA_F
- INISHE/INISH3 STRS_F
- INISHE/INISH3 STRS_GLOBAL
- INISHE/INISH3 AUX

Import of these cards also made optional with a check box in the solver options of the import browser. If the cards are not chosen to import they will be pushed to `.hmx` file. The `.hmx` file will be saved in import file location and attached as an include file with skip option.

Resolved Issues

- /PROP/TYPE19 referred in /PROP/TYPE17 lost during import is fixed.
- Error in exporting all Includes from the Include browser is fixed
- Read names completely when it has character "\"
- Duplicate /BEGIN card with import and include option is fixed
- Issue with reading /INTER/TYPE21 is fixed
- Reader failure while reading /RLINK is fixed
- Issue with reading /INTER/TYPE11 as per RADIOSS 110 format
- Transformation manager error when applying /TRANSFORM/SCA is fixed
- Line 1 repetition in export of /MAT/LAW15 is fixed

Back to top

11.0.130 Release Notes - HyperMesh

Solver Browser

The Solver Browser Show/Hide/Isolate/Review functions were updated for better performance. Review will display them in its color, if a color has been defined. If not, the display is red.

INISHE/EPSP and INISHE/EPSP_F

Support of /INISHE/EPSP and /INISHE/EPSP_F keywords has been added. Reading of this keyword is

made optional through solver options in the Import Browser. By default this keyword will be read into HyperMesh.

New Keywords

- /MONVOL/FVMBAG1
- /SENSOR/RBODY
- /SENSOR/SECT

Keywords Updated

Keywords updated based on RADIOSS solver update 11.0.201:

- /INTER/TYPE21
- /INTER/TYPE20
- /INTER/TYPE11
- /SENSOR/RWAL
- /SENSOR/INTER
- /DEF_SHELL
- /PROP/SHELL
- /PROP/SH_COMP
- /PROP/SH_ORTHO
- /PROP/SH_SANDW
- /EOS/TILLOTSON
- /EOS/GRUNEISEN
- /EOS/POLYNOMIAL

Resolved Issues

- Duplicate names appended with ".#" will be removed during export.
- The Title for /ADMAS keyword now supports up to 100 characters.
- /TRANSFORM is once again appearing in the Solver Browser.

Back to top

11.0.130 Release Notes - HyperView/ HyperGraph

New Reader Options

Introduced new reader options to improve the performance of commonly performed math operations:

- Strain Tensor with Prestrain removed – Subtracts the strain Tensor of first step from all time steps for all elements.
- Effective plastic strain with Prestrain removed – Subtracts the effective plastic strain of first step from all time steps for all elements.

- Maximum effective plastic strain over the layers – Gets the maximum effective plastic strain over the layer/thickness of shell elements.
- Maximum effective plastic strain over the layers with Prestrain removed – Subtracts the effective plastic strain of first step from all time steps then gets the maximum of over the layer/thickness of shell elements.

Resolved Issues - HyperView

- Handle PART and RBODY keyword with same ID

Resolved Issues - HyperGraph

- Missing time data in large files larger than 1GB size is now fixed.
- Mismatch in curve values between Windows and Linux platform for sequence of time history files (T01, T02, …) is now fixed.

Back to top

11.0.120 Release Notes - HyperMesh

Keyword Support

- Support of RADIOSS 11.0 – 201 keywords /EOS/PUFF, /EOS/SESAME, /FAIL/TAB, /PROP/TYPE26, and /PROP/SPR_TAB.
- Support of CNODE.

Resolved Issues

- Rigid body update fixed to avoid loss of selected nodes and potential crash.
- Import reader fixed to handle custom import of contact surfaces.

Back to top

11.0.120 Release Notes - HyperView/HyperGraph

HVTrans

HV Trans updated to handle model files that has duplicate name for components.

Back to top

11.0.101 Release Notes - HyperMesh

Transformations

Transformations defined in the model by default will be turned on during import.

Export Tab Enhancements

New checkbox has been introduced in the Export tab to avoid exporting any HM comment into the deck.

ACCEL

ACCEL keyword moved from HyperMesh entity Property to Sensor.

Newly Supported Keywords

- Newly supported Sensor keywords include /GAUGE, /SENSOR/GAUGE, and /TH/GAUGE.
- Newly supported ALE keywords include /INICONT, /INIVOL, /EOS_GRUNEISEN, /EOS_POLYNOMIAL, and /EOS_TILLOTSON
- Newly supported Failure keywords include /FAIL/XFEM_FLD, /FAIL/XFEM_TBUTC, and /FAIL/XFEM_JOHNS
- Newly supported Material keywords include /MAT/LAW69, /MAT/LAW74, and /MAT/LAW78

Updated Keywords

Supported keywords updated to RADIOSS 110 include:

- /MONVOL/FVMBAG (Airbag)
- /INTER/TYPE5 (Interface)
- /INTER/TYPE20 (Interface)
- /INTER/TYPE21 (Interface)
- /MAT/LAW4 (Materials)
- /MAT/LAW19 (Materials)
- /MAT/LAW42 (Materials)
- /MAT/LAW58 (Materials)

ID Pools

ID pools were added to the sensor entity to have duplicate IDs among its mapped keywords in RADIOSS (Block) UP/ACCEL, /SENSOR and /GAUGE.

Resolved Issues

- HyperMesh can read deck that have /UNIT keywords with combination of string and value
- ADMASS Manager performance has been improved for SPH models
- RBODY connectivity update behavior is updated to preserve the mass and inertia values
- Error in export of /PROP/TSH_COMP is fixed
- Error and crash in updating RBODY with more than 10,000 nodes is fixed

Back to top

11.0.101 Release Notes - HyperView/HyperGraph

Pyramid Elements

Support of pyramid elements has been added.

Creation of Section Cuts

Creation of section cuts based on /SECT keyword in the input deck when loaded as a model file.

Resolved Issues

- Error in VonMises strain fixed.
- Weird names for rigid elements and RWALL that appear in the HyperView Browser is fixed.
- Rigid elements appearing as beam elements in the HyperView Browser is fixed. Also, rigid elements by default are masked upon loading the file in HyperView.
- Added capability to handle assemblies that appear in the results file without any ID.

Back to top

Highlights of the RADIOSS & OptiStruct Interface 12.0 Release

- New Export Solver Option: Auto-adjust RBE3 0.0 weight factor
- Gasket Elements Support
- Solver Cards Supported Updates
- New OP2 Reader Option: Deformed Shape: Allow all nodal vector result types
- OP2 Format Results File
- Resolved Issues

Contents

11.0.130 Release Notes

NSM Mass Connectors

New Enhancements to HM-MBD/OS-MBD

New Material Support and Updates

Added Parameters to PARAM

Set Enhancements

New SMEARZO Options

Resolved Issues

New Reader Options

Resolved Issues (HyperView)

Resolved Issues (HyperGraph)

11.0.120 Release Notes

Keyword Support

Resolved Issues

HVTrans

11.0.101 Release Notes

Transformations

Export Tab Enhancements

ACCEL

Newly Supported Keywords

Updated Keywords

ID Pools

Resolved Issues

Pyramid Elements

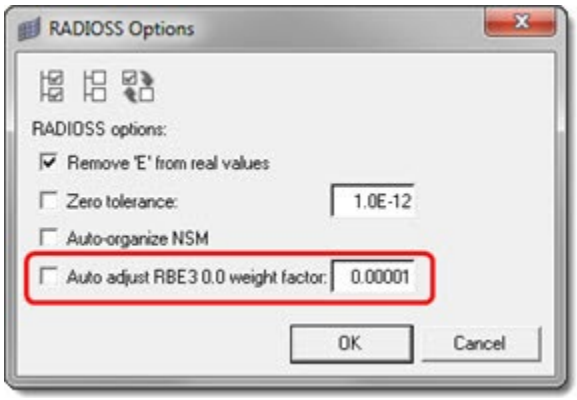
Creation of Section Cuts

Resolved Issues

12.0 Release Notes

New Export Solver Option: Auto-adjust RBE3 0.0 weight factor

When turned ON, HyperMesh will change any 0.0 weight factors on RBE3 elements to the specified small value. This adjustment is done in the exported solver deck only, and not in the HyperMesh model. This option is OFF by default.



Gasket Elements Support

Added support for the new RADIOSS (Bulk) gasket elements:

- CGASK6, CGASK8: New element types.
- MGASK: New material.
- PGASK: New property.
- Solid element normal review and editing now available for gasket elements.
  - Element normal direction review / reverse available via the Tool > Normals panel.
  - Advanced 3D normal direction control available via the 2D > Composites > Element Normals panel

---

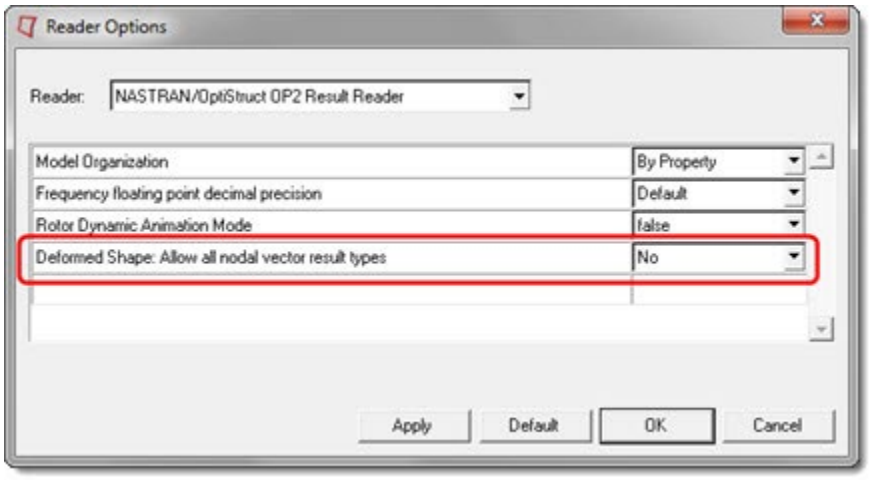
Solver Cards Supported Updates

- ASSIGN: STARTER and ENGINE options added.
- SOLVTYP: New AUTO option added.
- PFMODE, PFPANEL, PFGRID: Now available as both global and subcase-specific entries.
- CONTPRM: New IGNORE option added.
- MATX42: Removed obsolete ISTR option.
- MAT4: New CP field added.
- MAT5: New CP field added.

- PGAPHT: New extension card inside PGAP
- ERP: New Output Request.
- ERPPNL: New entity set type
- DOPTPRM: New REMESH and OPTMETH options added
- PARAM updates:
  - ERPC
  - ERPRLF
  - ERPRHO
  - ERPREFDB
  - RHOCP
  - AMSESLM
  - LFREQ
  - GE_MOD
  - DUPTOL

New OP2 Reader Option: Deformed Shape: Allow all nodal vector result types

When switched to YES, this option allows all available nodal vector result types in the OP2 file to be used in the Deformed Shape panel. When switched to NO (default), the Deformed Shape panel will automatically filter result type selection according to analysis type.



OP2 Format Results File

HyperView now supports importing of CONM2 elements from OP2 format results file.

Resolved Issues

HyperMesh interfacing resolved issues:

- NSML1 mass incorrectly distributed amongst 2D elements and properties issue has been corrected. This fix includes corrections to mass calc panel and summary templates.
- Pressure direction miss-assignment when importing PLOAD cards from separate files into a

current model has been corrected.

- DRESP2 cards containing a large number of DTABLE entries missing after export issue has been corrected.
- TABLEx export in Long Format missing second line (empty) issue has been fixed.
- SID of setdof inside PFMODE with type=STRUCTURE now requires SET of type=GRIDC only.
- ACMODL 2nd line missing when it only contains MAXSGRID parameter issue has been fixed.
- PCOMPP export in Long Format with miss-aligned Z0 issue has been fixed.
- TABLED missing when using Export > Displayed option issue has been fixed.
- DAREA/DPHASE on SPOINTS not importing properly has been fixed.
- PEAKOUT option exporting as PEAK inside several output requests issue has been fixed.

HyperView interfacing resolved issues:

- An issue when plotting SDISP in results files containing both DISP and SDISP results has been resolved.

Back to top

11.0.130 Release Notes - HyperMesh

NSM Mass Connectors

Added property type to NSM mass connectors. It is available for creation and FE absorption.

New Enhancements to HM-MBD/OS-MBD

- CMSPDP(E/T/C): New spring elements
- CMBUSH: New spring element
- CMBUSH(E/C/T): New spring elements
- MBVAR: New load collector
- MBREQ, MBREQE, and MBQEQM: New load collectors
- REQUEST: GLOBAL_CASE_CONTROL, MBD loadstep

New Material Support and Updates

- MAT9ORT
- MATX21 as an extension to MAT1
- MATX25 as an extension to MAT8
- MATX43 as an extension to MAT8
- MATX60 as an extension to MAT1
- MATX68 as an extension to MAT9ORT
- New option for MATS1 [mixed and kinematic hardening] HR.

Added Parameters to PARAM

- CMSDIRM
- CMSOFST
- CNTNLSUB
- COEFFC
- CSTEVAL
- DISIFMCK
- EHD
- EXCOUT
- LMSOUT
- MAXDAMP
- NPRBAR
- NPRBE2
- NPRGDE
- PLIGEXT
- RSPLICOR
- SEP1XOVR
- SEPLOT
- SIMPACK
- SMDISP
- SRCOMPS
- TOLRSC
- TPS

Set Enhancement

- PLOTEL

New SMEARZ0 Option

- New SMEARZ0 option added to PCOMP/PCOMPG/STACK

Resolved Issues

Coordinate systems without HM comments now organized into one system collector instead of one collector per coordinate system. This provides a major import speed improvement for models with large number of coordinate systems.

- Duplicate names appended with ".#" will be removed during export.
- The Title for /ADMAS keyword now supports up to 100 characters.
- /TRANSFORM is once again appearing in the Solver Browser.

Back to top

11.0.130 Release Notes - HyperView/ HyperGraph

New Reader Options

Introduced new reader options to improve the performance of commonly performed math operations:

- Strain Tensor with Prestrain removed – Subtracts the strain Tensor of first step from all time steps for all elements.
- Effective plastic strain with Prestrain removed – Subtracts the effective plastic strain of first step from all time steps for all elements.
- Maximum effective plastic strain over the layers – Gets the maximum effective plastic strain over the layer/thickness of shell elements.
- Maximum effective plastic strain over the layers with Prestrain removed – Subtracts the effective plastic strain of first step from all time steps then gets the maximum of over the layer/thickness of shell elements.

Resolved Issues - HyperView

- Handle PART and RBODY keyword with same ID

Resolved Issues - HyperGraph

- Missing time data in large files larger than 1GB size is now fixed.
- Mismatch in curve values between Windows and Linux platform for sequence of time history files (T01, T02, …) is now fixed.

Back to top

11.0.120 Release Notes - HyperMesh

Keyword Support

- Support of RADIOSS 11.0 – 201 keywords /EOS/PUFF, /EOS/SESAME, /FAIL/TAB, /PROP/TYPE26, and /PROP/SPR_TAB.
- Support of CNODE.

Resolved Issues

- Rigid body update fixed to avoid loss of selected nodes and potential crash.
- Import reader fixed to handle custom import of contact surfaces.

Back to top

11.0.120 Release Notes - HyperView/HyperGraph

HVTrans

HV Trans updated to handle model files that has duplicate name for components.

Back to top

11.0.101 Release Notes - HyperMesh

Transformations

Transformations defined in the model by default will be turned on during import.

Export Tab Enhancements

New checkbox has been introduced in the Export tab to avoid exporting any HM comment into the deck.

ACCEL

ACCEL keyword moved from HyperMesh entity Property to Sensor.

Newly Supported Keywords

- Newly supported Sensor keywords include /GAUGE, /SENSOR/GAUGE, and /TH/GAUGE.
- Newly supported ALE keywords include /INICONT, /INIVOL, /EOS_GRUNEISEN, /EOS_POLYNOMIAL, and /EOS_TILLOTSON
- Newly supported Failure keywords include /FAIL/XFEM_FLD, /FAIL/XFEM_TBUTC, and /FAIL/XFEM_JOHNS
- Newly supported Material keywords include /MAT/LAW69, /MAT/LAW74, and /MAT/LAW78

Updated Keywords

Supported keywords updated to RADIOSS 110 include:

- /MONVOL/FVMBAG (Airbag)
- /INTER/TYPE5 (Interface)
- /INTER/TYPE20 (Interface)
- /INTER/TYPE21 (Interface)
- /MAT/LAW4 (Materials)
- /MAT/LAW19 (Materials)
- /MAT/LAW42 (Materials)
- /MAT/LAW58 (Materials)

ID Pools

ID pools were added to the sensor entity to have duplicate IDs among its mapped keywords in RADIOSS (Block) UP/ACCEL, /SENSOR and /GAUGE.

Resolved Issues

- HyperMesh can read deck that have /UNIT keywords with combination of string and value
- ADMASS Manager performance has been improved for SPH models
- RBODY connectivity update behavior is updated to preserve the mass and inertia values

- Error in export of /PROP/TSH_COMP fixed
- Error and crash in updating RBODY with more than 10,000 nodes is fixed

Back to top

11.0.101 Release Notes - HyperView/HyperGraph

Pyramid Elements

Support of pyramid elements has been added.

Creation of Section Cuts

Creation of section cuts based on /SECT keyword in the input deck when loaded as a model file.

Resolved Issues

- Error in von Mises strain fixed.
- Weird names for rigid elements and RWALL that appear in the HyperView Browser is fixed.
- Rigid elements appearing as beam elements in the HyperView Browser is fixed. Also, rigid elements by default are masked upon loading the file in HyperView.
- Added capability to handle assemblies that appear in the results file without any ID.

Back to top

Highlights of the Samcef Solver Interface 12.0 Release

- Parameter NEXCE

Contents

11.0.130 Release Notes

FE Input Reader

Hypotheses Enhancements

11.0.101 Release Notes

FE Input Reader

Generic Format

.FCT

Creating a Curve

PLASTIC

Contact Definition

Non Linear Analysis

Resolved Issues

12.0 Release Notes

Parameter NEXCE

Support the offset value in the shell and beam properties through the parameter NEXCE.

Back to top

11.0.130 Release Notes

FE Input Reader

FE Input Reader has been enhanced to be able to read the Samcef bank files and the files generated by HyperMesh. To generate a bank file (generic file format), use the BACON interface or the command line .SAUV BANQUE. After that, you will be able to import the new file generated.

Hypotheses Enhancements

Hypotheses have been updated to be in line with the default values of Samcef.

Back to top

11.0.101 Release Notes

FE Input Reader

- Using the FE Input Reader, you can now import a complete model created in HyperMesh. We support all the features that we support at the export.

Generic Format

- For the models created in SAMCEF, the generic format: banque file is now supported.

.FCT

- The macro command .FCT is now supported.

Creating a Curve

- To create a curve, you can now use the menu bar item XYPlots > Curve Editor.

PLASTIC

- The behavior PLASTIC is now supported. The plasticity is defined by a hardening function.

Contact Definition

- All the details for the contact definition are in the section HyperMesh > User's Guide > HyperMesh Entities & Solver Interfaces > Named Entities > Material.

Non Linear Analysis

- You can define a non linear analysis in the panel Analysis > LoadSteps. You need to create a loadstep and after that you can edit it to define the parameters for the non linear analysis.
- All the details for the definition of a linear or non linear subcase are in the section HyperMesh > User's guide > HyperMesh Entities & Solver Interfaces > Named Entities > Load Steps.
- The macro command .MCE MEAN/.MCC MEAN is now supported. To create this element, use the RBE3 panel.
- Now you can create a load in function of a curve. In the Constraint, Force, Moment or Pressure panels, click on magnitude and select curve, vector or curve, components.

Resolved Issues

- In previous versions, even if you modify the DOF during the creation of the RBE element, all the dof (CM 123456) we exported. Now the DOFs are correctly exported.

Back to top

Highlights of the Evolve 9.0 Release

A 3D conceptual design environment which empowers designers to swiftly capture and evolve forms using powerful NURBS modeling tools, an advanced construction history and photorealistic visualization.

Contents

New Features

User Interface/Workflow

Modeling Interaction

Rendering

Input/Output

Plugins

Localization

New Features

User Interface/Workflow

- Improved model shading and visualization options.
- New flag notifications for objects needing to be updated or repaired.
- New “smart delete” for hiding/un-hiding source objects when deleting items.
- Updated User Interface (UI) - new color scheme - new icons for better visual communication
- Editable IU Colors - added option for the user to edit all the colors in the UI in the Preferences Panel
- Preferences UI tab - new tab in Preferences panel to control UI
- Undo Buffer Toggle - new on/off checkbox - current status is indicated in application titlebar - no longer need to restart to take effect
- Modeling Tool Panel (MTP) - cleaner interface and new default dock position
- Layer On/Off toggle - consolidated the two icons to simplify the workflow
- Layer Lock toggle - the Layer Lock option now works as a toggle to turn the layer on and off
- Modeling window visualization modes - new icons - added zebra strips - improved environment shaders - independent control of edge lines, isoparms, or neither
- Camera icons Show/Hide option - ability to show or hide the camera icons in each of the modeling windows

- Improved surface shaders - ability to control color, shininess, reflectivity, and transparency per object
- Improved model tessellation - new tessellation calculation for higher quality visualization and increased speed for handling large files
- "Maximum" View Detail button - additional button added to the View Detail window for highest visual quality
- Curve Extract tools relocation - curve extract tool icons have been relocated to be more visually intuitive
- Curve selection filter hotkey - pressing the "1" key while selecting applies the curve selection filter
- Improved Snap tolerances - increased snap strength and tolerance
- Object naming - objects include modeling tool name and are uniquely numbered
- Flags for broken objects - in the event an object in Construction History is broken a flag is added to that object in the World Browser
- Copy/Paste - ability to retain Construction History, and ability to copy/paste between multiple sessions of sT
- Translation/Rotation/Scale persistent - transformation tools now stay active allowing multiple edits - spacebar and right-click are shortcuts added to close the tool
- Translation auto-activate - Translation manipulator is activated in the Instances tool and when an object is pasted
- 3D Connexion support - added support for the 3D Connexion devices
- User feedback option - added "Submit Feedback" in Help to report issues or requests

---

Modeling Interaction

- New Instance tool allows copied objects to be controlled from a source object
- Multiple object support for Trim, Boolean, and Intersect
- New Loft and Patch tools improve surface continuity control
- Improved Modeling tolerances - tolerances are adaptive for more robust geometry creation
- New Split Surface tool - allows a surface to be trimmed by another surface without altering the "tool" surface
- New Make Manifold - improved robustness
- Improved Blend Surface - added options for Isoparm constrants - added Start Over option to rebuilt the surface to maintain construction history
- Improved Polymesh tools - added Edge Loop option for better workflow
- Polymesh visual tessellation - added option to use the modelling view settings for creating the polymesh tessellation

---

Rendering

- Improved functionality for controlling multiple textures and materials per surface
- Added Keyshot support – files can be saved in .bip format and directly launched in Keyshot
- Interactive Rendering - formerly Realtime Rendering - renamed to be more descriptive
- Interactive Rendering for selected objects - added ability to perform Interactive rendering for selected objects

- Labels - simplifies ability to apply and control multiple textures/images to a single object
- Shadows for default lights - Shadows have been added to the default lights with the option to set hard or soft as the default
- New rendering shortcuts - added and modified keyboard shortcuts for the various rendering methods

---

Input/Output

- All file formats - updated to current versions
- .obj - added import and export capability for .obj files
- Improved H3D support - MBD - flex body - primitive support

Plugins

- SolidWorks - plugin updated to support SW 2008 – 2012

Localization

- Graphic User Interface – added Spanish and Portuguese

Back to top

Highlights of the Inspire 9.0 Release

Contents

New Features

- Sketch Lines, Rectangles, Circles and Arcs
- Constraint Sketch Geometry
- Boolean Operations
- Maximum Thickness Control
- Frequency Targets
- Faster Shape Exploration
- Perspective/Orthogonal Projections
- Align to Closest Axis
- Improved Geometry Capture
- Altair HyperWorks Integration
- 64 bit Executable
- Enable/Disable Rotate Constraints on Holes
- Japanese Version
- Tutorials

New Features

Sketch Lines, Rectangles, Circles and Arcs

Build simple design spaces by modifying or creating new solid parts using lines, rectangles, circles and arcs. Sketched geometry can be exported from Inspire in various formats including Parasolid, STEP, and IGES.

Constraint Sketch Geometry

Geometric relationships such as tangents, perpendicularity can be applied to sketches. When sketches are modified, constraints ensure the intended relationship between the sketch and entity.

Boolean Operations

New solid objects can be created by performing Boolean operations to combine, subtract or intersect sets of objects.

Maximum Thickness Control

Users can control wall thicknesses and the diameters of beam-like structures in a generated shape by specifying a minimum and/or maximum thickness.

Frequency Targets

Design concepts can be optimized to maximize or meet specified frequency targets.

Faster Shape Exploration

The shape exploration function is now controlled by a single material slider.

Perspective/Orthogonal Projections

Users can switch between perspective and orthogonal views for more precise sketching.

Align to Closest Axis

Models also can be aligned to the closest principal axis when oriented arbitrarily.

Improved Geometry Capture

Powerful new internal algorithm results in higher quality design concepts.

Altair HyperWorks Integration

Files generated by Inspire can be exported to HyperMesh for validation and advanced structural analyses.

---

64 bit Executable

A 64-bit executable is available for Windows 7, Vista and XP.

Enable/Disable Rotate Constraints on Holes

Rotations are now enabled for constraints located on holes.

Japanese Version

Inspire is available in native Japanese language.

Tutorials

New tutorials with videos highlighting new features have been added.

Back to top